Empire XPU
Manual



Version 9.2.2

(c) IMST GmbH 1998-2026

August 4, 2026

Contents

I. Software Administration 1

1. Installation on Windows PC 2
- 1.1. Requirements for PC Installation . . . 2
- 1.2. Installation Procedure . . . 2
- 1.3. Multiple version installations . . . 6
- 1.4. Silent installation on Windows . . . 6
- 1.5. Uninstall . . . 6

2. License Setup 7
- 2.1. License identification numbers . . . 7
- 2.2. Overview of Licensing Options . . . 8
- 2.3. Dongle licensing . . . 8
- 2.4. Request a License . . . 9
- 2.5. Install the License . . . 10
- 2.6. License File . . . 11

3. Starting Empire on Windows 13
- 3.1. Simple Test Procedure . . . 13
  - 3.1.1. License test . . . 13
  - 3.1.2. Graphic test . . . 13
  - 3.1.3. Compiler test . . . 15
  - 3.1.4. Plot library test . . . 15
  - 3.1.5. Visualization tool kit test . . . 16
- 3.2. Simulation from the Windows command line . . . 16
- 3.3. Graphic Renderer Information . . . 17

4. Floating License on Windows 18
- 4.1. Floating License File . . . 18
- 4.2. Installation on the Server . . . 19
  - 4.2.1. Delayed license checkout . . . 21
- 4.3. Installation on each Client . . . 21
- 4.4. License Diagnostic . . . 22

5. Remote Control 24
- 5.1. Remote Desktop . . . 24
  - 5.1.1. Windows Remote desktop . . . 24
  - 5.1.2. PCoIP . . . 25
- 5.2. Remote computing . . . 25
  - 5.2.1. Starting the remote service in EMPIRE XPU™ . . . 26
  - 5.2.2. Remote service Windows commands . . . 27
  - 5.2.3. Remote service Linux commands . . . 27
  - 5.2.4. Starting the remote service as a Windows service . . . 27
  - 5.2.5. Batch processing using IP addresses . . . 29

6. Multi-PC Solver 30

7. Virtual Model Inspection Module 32

8. Linux systems 33
8.1. Requirements . . . 33
8.2. Installation . . . 33
8.3. Silent installation on Linux . . . 34
8.4. Start Commands . . . 34
8.4.1. Background processing . . . 35
8.5. License Request . . . 35
8.6. Install the License . . . 35
8.7. Floating license . . . 35
8.7.1. Install Programs . . . 35
8.7.2. Check Interface Name . . . 36
8.7.3. Start License Server . . . 36
8.7.4. Delayed license checkout (Linux) . . . 37
8.8. Trouble Shooting . . . 37

9. Data Structure 39
9.1. Input Files . . . 39
9.1.1. File Recovery . . . 39
9.1.2. Compatibility . . . 39
9.2. Simulation Files . . . 39
9.3. Postprocessing Files . . . 40

10. High Performance Computing 41
10.1. Single machine consideration . . . 41
10.2. Multiple machine considerations . . . 42
10.2.1. Multiple jobs on different machines (job distribution) . . . 42
10.2.2. Single jobs on different machines (job sharing) . . . 42

11. Update & Support 44
11.1. Version Control . . . 44
11.2. Web Sites . . . 44
11.3. Contact Us . . . 44
11.4. Contact Local Support Teams . . . 45

II. Empire User Interface 46

12. Introduction 47
12.1. Top tool bars . . . 48
12.1.1. Menu items . . . 48
12.1.2. Application tabs . . . 52
12.1.3. Top left icons . . . 52
12.2. 3D Design Mode . . . 55
12.2.1. Grid . . . 55
12.2.2. Cursor . . . 57
12.2.3. Coordinate Systems . . . 57
12.2.4. Zoom . . . 59
12.2.5. Pan . . . 59
12.2.6. Rotate . . . 59
12.2.7. Bottom line . . . 60
12.2.8. 3D Outline View . . . 60

12.3. 2D Design Mode . . . 60
12.3.1. Background . . . 61
12.3.2. Zoom . . . 61
12.3.3. Pan . . . 61
12.3.4. Origin, WCS . . . 61
12.3.5. Simulation domain . . . 61
12.3.6. Bottom line . . . 62
12.3.7. 2D Outline View . . . 63
12.3.8. Toggle Groups in 2D Mode . . . 63
12.4. 3D Results Mode . . . 63
12.4.1. Background . . . 63
12.4.2. Zoom . . . 63
12.4.3. Pan . . . 63
12.4.4. Rotate . . . 64
12.4.5. Display Buttons . . . 64
12.4.6. Context Menu . . . 65
12.5. Operation buttons . . . 66
12.6. Side menu / Navigation tree . . . 67

13. Operators, Selections and Shortcuts **69**
13.1. Operators . . . 69
13.2. Object types . . . 69
13.3. Select objects . . . 70
13.4. Handles . . . 70
13.5. Deselect objects . . . 70
13.6. Shortcuts . . . 71

14. Groups **72**
14.1. Control for all groups . . . 73
14.2. Control for single groups . . . 74

15. Object creation **76**
15.1. Basic Objects . . . 76
15.1.1. Box . . . 77
15.1.2. Circular Cylinder . . . 77
15.1.3. Polygon . . . 78
15.1.4. Strip . . . 80
15.2. Object Library . . . 81
15.2.1. 3D Solid . . . 82
15.2.2. 3D Surface . . . 84
15.2.3. 3D Wire . . . 86
15.2.4. 2D Extruded . . . 86
15.2.5. Coax . . . 88
15.2.6. MSL Planar . . . 91
15.2.7. CPW Planar . . . 91
15.2.8. SMDs . . . 93
15.2.9. Substructure . . . 94
15.2.10. Layout . . . 98
15.2.11. Conformal . . . 99
15.2.12. User . . . 100

16. Object modification **101**
16.1. Move Operations . . . 101
16.1.1. Move Object . . . 101

16.1.2. Move Object to Group . . . 101
16.1.3. Parametric Move . . . 102
16.1.4. Mirror . . . 102
16.1.5. Rotate . . . 102
16.1.6. Align . . . 103
16.1.7. Transform via LCS . . . 103
16.2. Copy Operations . . . 103
16.2.1. Copy Object . . . 103
16.2.2. Copy to Current Group . . . 104
16.2.3. Array Copy . . . 104
16.2.4. Copy & Mirror . . . 105
16.2.5. Copy & Rotate . . . 105
16.3. Stretch Operations . . . 105
16.3.1. Stretch Object . . . 105
16.3.2. Parametric Stretch . . . 106
16.3.3. Scale . . . 106
16.3.4. Oversize . . . 107
16.3.5. Rasterize Polygons . . . 107
16.4. Change operations . . . 107
16.4.1. Set radius . . . 107
16.4.2. Convert Boxes . . . 107
16.4.3. Convert Polygons . . . 108
16.4.4. Convert Solid . . . 108
16.5. Boolean operations . . . 108

17. Import and Export **110**
17.1. Import . . . 110
17.1.1. Advanced . . . 110
17.1.2. Empire . . . 111
17.1.3. EDA . . . 111
17.1.4. 2D Layout Data . . . 112
17.1.5. 3D Solid Data . . . 113
17.1.6. CST-Projects . . . 116
17.1.7. HFSS-Projects . . . 118
17.1.8. Data - Import Touchstone . . . 119
17.2. Export . . . 119
17.2.1. Advanced . . . 119
17.2.2. 2D Layout . . . 120
17.2.3. 3D Solids . . . 121
17.2.4. Empire Part . . . 122
17.2.5. Read Protected File . . . 122
17.2.6. Export Image . . . 122

18. Object properties **123**
18.1. Priorities . . . 123
18.2. Material Property . . . 124
18.2.1. Dielectric . . . 124
18.2.2. Conductor . . . 125
18.2.3. Absorber . . . 127
18.2.4. Advanced - Material Object . . . 129
18.2.5. Advanced - Debye Material . . . 130
18.2.6. Advanced - Drude Plasma . . . 134
18.2.7. Advanced - Meta Material . . . 134
18.2.8. Advanced - Gabriel Material . . . 136

18.2.9. Advanced - Conformal Dielectrics . . . 137
18.2.10. Advanced - Metal Sheet Stack . . . 140
18.2.11. Advanced - Material Scripts . . . 140
18.3. Circuit Property . . . 140
18.3.1. Volume objects . . . 140
18.3.2. Advanced - Surface objects . . . 141
18.3.3. Advanced - Line objects . . . 142
18.4. Thermal Property . . . 142
18.5. Mesh Property . . . 142
18.5.1. Mesh Hint . . . 142
18.5.2. Mesh Accuracy and Quality Check . . . 144
18.5.3. Ignore Object . . . 144
18.5.4. Discretize Solid Surface only . . . 144
18.5.5. Discretize Solid Interior only . . . 144
18.5.6. Discretize Solids as wires . . . 144
18.6. Advanced Property . . . 144
18.6.1. Voxel model . . . 144
18.6.2. Field Storage & Display . . . 146
18.6.3. Measurement Definition . . . 146
18.6.4. Excitation Definition . . . 146
18.6.5. Layout Export hints . . . 147
18.6.6. No Dispersion Correction . . . 147
18.6.7. Experimental . . . 147
18.6.8. User defined . . . 147

19. Field Monitors **148**
19.1. EM Field . . . 148
19.1.1. Storage Options . . . 149
19.1.2. 3D Plot Options . . . 151
19.1.3. 3D Result Options . . . 156
19.2. Waveguide Port Modes . . . 156
19.3. EM Far Field . . . 159
19.3.1. Storage Options . . . 159
19.3.2. Processing Setup . . . 161
19.3.3. Mimo Correlation Setup . . . 164
19.3.4. Farfield Superposition . . . 165
19.3.5. Plot Options . . . 167
19.3.6. 3D Result Options . . . 167
19.4. EM Probes . . . 168
19.4.1. Field Line . . . 169
19.4.2. Point Field Probe (Frequency Domain) . . . 171
19.4.3. Point Field Probe (Time Domain) . . . 173
19.5. Thermal Field Monitor . . . 173
19.5.1. Thermal Display . . . 173
19.5.2. Thermal Point Probe . . . 176

20. Sources **177**
20.1. Ports . . . 177
20.1.1. Port Setup Wizard . . . 179
20.1.2. QTEM Ports . . . 180
20.1.3. Lumped Ports . . . 185
20.1.4. Feedline Ports . . . 189
20.1.5. TE-TM Waveguide Ports . . . 198

20.2. Volume Sources . . . 202
20.2.1. Plane Wave . . . 202
20.2.2. Field Source (Resizable) . . . 204
20.2.3. Field Source (Fixed Size) . . . 205
20.3. Thermal Source . . . 206
20.3.1. Thermal Source from EM loss . . . 207
20.3.2. Thermal Power Source . . . 207
20.4. EM Twin . . . 208

21. Variables **209**
21.1. Create Variables . . . 210
21.2. Control Variables . . . 210
21.3. Equations . . . 212
21.4. Sweep Simulation . . . 214
21.5. Sweep Results . . . 214

22. Meshing and Simulation Domain **215**
22.1. Automatic Meshing . . . 216
22.1.1. Mesh Creation . . . 216
22.1.2. Simulation Domain . . . 216
22.1.3. General Mesh Options . . . 217
22.1.4. Mesh Mode and Mesh Hint . . . 218
22.1.5. Simulation Box . . . 219
22.1.6. Boundary Distance . . . 219
22.1.7. Advanced Mesh Setup . . . 219
22.1.8. Group Meshing . . . 221
22.1.9. Meshing Algorithm . . . 221
22.2. Manual Meshing . . . 223
22.2.1. Marking intervals . . . 224
22.2.2. Generic discretization . . . 224
22.2.3. Delete Lines . . . 224
22.2.4. Add lines . . . 225
22.2.5. Edit Lines . . . 225
22.2.6. Port discretization . . . 225

23. Simulation Setup **226**
23.1. General Setup . . . 226
23.2. EM Setup . . . 228
23.2.1. Frequency . . . 228
23.2.2. Resonance Estimation . . . 229
23.2.3. Loss Calculation . . . 229
23.2.4. Accuracy . . . 231
23.2.5. Enhancement . . . 231
23.2.6. Perfect Geometry Approximation (PGA) . . . 231
23.2.7. Time Domain Reflectometry (TDR) . . . 232
23.2.8. Misc . . . 232
23.2.9. Gabriel Object Options . . . 232
23.3. Thermal . . . 232
23.3.1. Thermal . . . 233
23.3.2. Surface . . . 233
23.3.3. SAR/ACD . . . 233
23.4. Mesh . . . 234
23.5. Boundary . . . 234
23.5.1. Boundary conditions . . . 234

23.5.2. Boundary Distance . . . 235
23.6. Excitation . . . 235
23.6.1. Excitation Setup . . . 236
23.6.2. Pulse shape . . . 236
23.7. End Criteria . . . 237
23.7.1. End Criteria Setup . . . 238
23.8. Equations . . . 238
23.8.1. Symbolic Equation . . . 239
23.8.2. Calculation . . . 240
23.8.3. File Copy . . . 240
23.8.4. Touchstone File . . . 241
23.8.5. Field Superposition . . . 241
23.8.6. Far Field Calculation . . . 241
23.9. Port Setup . . . 242
23.10Multi PC / On-disc Simulation Setup . . . 242
23.10.1Slave Host Setup . . . 243
23.11Options . . . 243

24. Simulation **246**
24.1. Preprocessing . . . 246
24.2. Quick Start . . . 246
24.3. Simulation Control . . . 247
24.3.1. Simulation Log . . . 247
24.3.2. Simulation Plot . . . 248
24.3.3. Simulation Actions . . . 248
24.4. Basic Simulation . . . 249
24.5. Advanced Simulation . . . 249
24.6. Batch processing . . . 252
24.7. Parameter Sweep . . . 253
24.8. Mesh Accuracy and Quality Check . . . 254
24.8.1. Hard XYZ-Mesh Refinement . . . 255
24.8.2. Automesh Expert-Mesh Refinement . . . 256
24.9. Optimization . . . 256
24.9.1. Optimization Control . . . 257
24.9.2. Optimization Configuration . . . 257
24.9.3. Optimization Goal . . . 258
24.9.4. Optimization Goal Examples . . . 260
24.10Remote Processing . . . 261
24.11Post Processing . . . 262

25. Results **263**
25.1. 2D Results set up . . . 263
25.1.1. Button Bar . . . 264
25.1.2. Graph Frame . . . 265
25.1.3. 2D Curve Display Setup - Plot Type . . . 267
25.1.4. 2D Curve Display Setup - Plot Format . . . 269
25.1.5. 2D Curve Display Setup - Results . . . 270
25.1.6. 2D Curve Display Setup - Filter . . . 271
25.1.7. 2D Curve Display Setup - Marker . . . 271
25.1.8. 2D Curve Display Setup - Masks . . . 272
25.1.9. Variable Sweep & Curve Setup . . . 274
25.2. 3D Result Display . . . 274
25.2.1. Near field . . . 274
25.2.2. Far field . . . 274

25.3. Result Files . . . 275
25.3.1. File location . . . 275
25.3.2. File processing . . . 276
25.3.3. File formats . . . 277

26. Options **278**
26.1. Editor Option Menu . . . 278
26.1.1. 2D Design . . . 278
26.1.2. 2D Design Advanced . . . 280
26.1.3. 3D Design . . . 280
26.1.4. 3D Design Advanced . . . 281
26.1.5. Import . . . 282
26.1.6. Export . . . 284
26.1.7. Export Advanced . . . 285
26.1.8. Components . . . 286
26.1.9. Conversion . . . 288
26.1.10. Design . . . 288
26.1.11. Dimensioning . . . 289
26.1.12. Preferences . . . 289
26.2. 2D Result Options . . . 290
26.3. 3D Result Options . . . 291
26.3.1. Basics . . . 293
26.3.2. Preferences . . . 294
26.3.3. Lighting . . . 294

27. Project Wizard **295**
27.1. New Project . . . 295
27.2. Open Project . . . 295
27.3. Examples . . . 297
27.3.1. Example quick reference guide . . . 297
27.4. Templates - Transmission Lines & Waveguides . . . 297
27.4.1. Template quick reference guide . . . 298
27.4.2. Microstrip Line . . . 298
27.4.3. Strip Line . . . 299
27.4.4. Coplanar Waveguide . . . 299
27.4.5. Coaxial Waveguide . . . 300
27.4.6. Rectangular Wave Guides . . . 300
27.4.7. Plate Plane Wave . . . 300
27.4.8. Conformal . . . 300
27.5. Templates - Coupled Transmission Lines . . . 301
27.6. Templates - SMD chip components . . . 301
27.7. Templates - RLC . . . 301
27.7.1. Rectangular Inductor . . . 301
27.7.2. Interdigital Capacitor . . . 301
27.7.3. Meander Resistor . . . 301
27.8. Templates - Antennas . . . 301
27.8.1. Dipole . . . 301
27.8.2. Monopole . . . 302
27.8.3. Microstrip Antenna . . . 302
27.8.4. Inverted-F Antenna . . . 302
27.8.5. Pyramidal Horn Antenna . . . 303
27.8.6. Mobile phone . . . 303
27.8.7. BiQuad . . . 303
27.8.8. Patch Antenna Array . . . 303

27.9. Templates - Filters . . . 303
27.9.1. Coupled Lines . . . 304
27.9.2. Coupled Hairpin . . . 304
27.9.3. Tapped Hairpin . . . 304
27.10Templates - Thermal . . . 304
27.11Templates - Environments . . . 304
27.12Tutorials . . . 304
27.12.1Tutorial quick reference guide . . . 304

28. Scripting **306**
28.1. User defined elements . . . 306
28.1.1. User defined polygons . . . 306
28.1.2. User defined library element . . . 307
28.2. Material distributions . . . 308
28.2.1. Material Script . . . 308
28.3. Scripted input, setups and environments . . . 310
28.3.1. Set Simulation Parameters . . . 310
28.3.2. Create Layer Stack . . . 311
28.3.3. Create objects . . . 311
28.3.4. Create and modify objects . . . 311
28.3.5. Create ports and monitors . . . 312
28.4. Scripted Simulations . . . 313
28.5. Scripted Postprocessing . . . 314
28.6. Supported Functions . . . 314
28.6.1. Functions in scripting.py . . . 315
28.6.2. Functions in gmf.py . . . 318

III. Application Guide **326**

29. EM Simulation **328**
29.1. Basic Applications . . . 328
29.1.1. ACD calculation (Averaged Current Density) . . . 329
29.1.2. Active impedance calculation . . . 329
29.1.3. Capacitance calculation . . . 329
29.1.4. Circular Polarization (Antennas, Waveguides) . . . 330
29.1.5. Efficiency calculation (Antennas) . . . 331
29.1.6. Far field animation . . . 331
29.1.7. Far field plot . . . 332
29.1.8. Far field recording . . . 332
29.1.9. Gain calculation (Antennas) . . . 333
29.1.10Impedance calculation . . . 333
29.1.11Inductance calculation . . . 334
29.1.12Low frequency algorithm . . . 334
29.1.13Multiple port excitation (independent) . . . 335
29.1.14Near field animation . . . 335
29.1.15Near field plot . . . 336
29.1.16Near field recording . . . 336
29.1.17Optimization . . . 337
29.1.18Parameter Sweep . . . 337
29.1.19Phase center calculation (Antennas) . . . 338
29.1.20Plane wave excitation . . . 338
29.1.21Port creation (absorbing) . . . 338
29.1.22Port creation (concentrated) . . . 339

29.1.23 Quality factor calculation (Inductance) . . . 339
29.1.24 RCS calculation . . . 340
29.1.25 SAR calculation . . . 340
29.1.26 S-Parameter calculation . . . 341
29.1.27 Touchstone file (postprocessing) . . . 341
29.1.28 User equations (postprocessing) . . . 342
29.1.29 Waveguide excitation (hollow) . . . 342
29.2. Application Notes . . . 345
29.2.1. Batch Processing . . . 345
29.2.2. CAD Import . . . 347
29.2.3. Conformal Toolbox . . . 349
29.2.4. Conformal-Dielectric . . . 351
29.2.5. CST Import . . . 353
29.2.6. Signal Estimation . . . 355
29.2.7. Far-Field Recording . . . 357
29.2.8. Far-Field Display . . . 359
29.2.9. Filter Template . . . 361
29.2.10 HFSS Import . . . 363
29.2.11 Layout Export . . . 365
29.2.12 Layout Import . . . 367
29.2.13 Mesh Setup . . . 369
29.2.14 Multilayer Meshing . . . 371
29.2.15 MultiPC Processing . . . 373
29.2.16 Optimizer . . . 375
29.2.17 Parameter Sweep . . . 377
29.2.18 Protected Part . . . 379
29.2.19 Remote Processing . . . 381
29.2.20 Empire Server v2 . . . 383
29.2.21 TDR Simulation . . . 385
29.2.22 Thermal Simulation . . . 387
29.2.23 Via Fence . . . 389

30. Circuit Simulation **390**
30.1. Circuit Simulation . . . 390
30.2. EM - Circuit Co-Simulation . . . 390
30.3. EM - Circuit Co-Simulation with Far field superposition . . . 392
30.4. Circuit Library . . . 392
30.4.1. General . . . 394
30.4.2. Lumped Elements . . . 394
30.4.3. Transmission Lines . . . 394
30.4.4. Data Elements . . . 394
30.4.5. Coupler . . . 394
30.5. Circuit Simulation Setup . . . 395
30.5.1. Circuit Simulation Configuration . . . 395
30.5.2. Circuit Editor Options . . . 395

31. Thermal Simulation **396**
31.1. Thermal Simulation Setup . . . 396
31.2. Thermal Solver . . . 397

32. Body Model Simulation **398**
32.1. Load Voxel File . . . 398
32.1.1. Import Voxel File . . . 399

IV. Command and Setup Reference 400

33. Actions 401
33.1. Boolean . . . 401
33.2. Clear . . . 402
33.3. Create . . . 402
33.4. Delete . . . 407
33.5. Design . . . 407
33.6. Display . . . 408
33.7. Edit . . . 409
33.8. File . . . 412
33.9. Group . . . 413
33.10 Mesh . . . 413
33.11 Misc . . . 414
33.12 Mode . . . 415
33.13 Origin . . . 416
33.14 Polygons . . . 416
33.15 Select . . . 417
33.16 Transform . . . 418
33.17 View . . . 421
33.18 Advanced . . . 422
33.18.1 Advanced/Arrows . . . 422
33.18.2 Advanced/Boxes . . . 424
33.18.3 Advanced/Create . . . 424
33.18.4 Advanced/Design Check . . . 425
33.18.5 Advanced/Edit . . . 428
33.18.6 Advanced/Group . . . 430
33.18.7 Advanced/LibraryElements . . . 430
33.18.8 Advanced/Linpolys & Bond Wires . . . 431
33.18.9 Advanced/Mesh . . . 432
33.18.10 Advanced/Parameters . . . 439
33.18.11 Advanced/Polygons . . . 439
33.18.12 Advanced/Properties . . . 442
33.18.13 Advanced/Scripting . . . 443
33.18.14 Advanced/Select . . . 443
33.18.15 Advanced/Solid . . . 444
33.18.16 Advanced/Transform . . . 444

34. Editor Options 447
34.1. 2D Design . . . 447
34.1.1. 2D Design/Arrow . . . 447
34.1.2. 2D Design/Component Snap . . . 447
34.1.3. 2D Design/Cursor . . . 448
34.1.4. 2D Design/General Snap . . . 449
34.1.5. 2D Design/Point . . . 450
34.1.6. 2D Design/View . . . 450
34.2. 2D Graphs . . . 451
34.3. 3D Design . . . 451
34.4. 3D Display . . . 452
34.4.1. 3D Display/Field Monitor Animation . . . 452
34.4.2. 3D Display/Lights . . . 452
34.4.3. 3D Display/View . . . 454
34.5. Components . . . 455
34.5.1. Components/Bond Wires . . . 455

34.5.2. Components/Polygons . . . 455
34.5.3. Components/Solids . . . 457
34.5.4. Components/Via Fence . . . 459
34.6. Design . . . 461
34.7. Dimensioning . . . 462
34.8. Export . . . 463
34.8.1. Export/2D Image . . . 463
34.8.2. Export/2D Layout . . . 464
34.8.3. Export/3D Image . . . 465
34.8.4. Export/3D Solids . . . 465
34.8.5. Export/Advanced . . . 466
34.9. Import . . . 468
34.9.1. Import/2D Layout . . . 468
34.9.2. Import/3D Solids . . . 470
34.9.3. Import/Empire . . . 471
34.9.4. Import/Advanced . . . 471
34.10Preferences . . . 472

35. Autodisc Options **473**
35.1. General . . . 473
35.2. Advanced . . . 473
35.2.1. Advanced/Options . . . 473
35.2.2. Advanced/User Setup . . . 474
35.3. Axis Setup . . . 477
35.4. Simulation Box . . . 479
35.4.1. Simulation Box/X . . . 479
35.4.2. Simulation Box/Y . . . 479
35.4.3. Simulation Box/Z . . . 479

36. Simulation Options **481**
36.1. Accuracy . . . 481
36.2. Additional Space . . . 482
36.3. Advanced . . . 483
36.3.1. Advanced/Gabriel Object Options . . . 483
36.4. Boundary Conditions . . . 483
36.5. Frequency . . . 485
36.6. General . . . 486
36.7. Loss Calculation . . . 487
36.8. Meshing1 . . . 488
36.9. Meshing2 . . . 488
36.10Port Setup . . . 489
36.11Thermal . . . 489
36.11.1Thermal/SAR/ACD . . . 490
36.11.2Thermal/Surface . . . 490

V. Error Messages **492**

37. GUI Messages **494**
37.1. General . . . 494
37.2. Fields and Animation . . . 494
37.3. Objects . . . 495
37.4. Import and Export . . . 497

38. Compilation Messages **499**

39. Kernel Messages **501**

VI. Theoretical Background 503

40. General Description **504**
40.1. Simulation Time . . . 504
40.2. Performance . . . 504
40.3. Number of Timesteps . . . 505
40.4. Number of Cells . . . 505
40.5. Simulation Memory . . . 505
40.6. Simulation Assumptions . . . 505
40.7. The Yee algorithm . . . 506

41. Discretization **509**
41.1. Meshing . . . 509

42. Boundary Conditions **510**
42.1. Electric Wall . . . 510
42.2. Magnetic Wall . . . 510
42.3. Absorbing Boundaries . . . 511
42.3.1. Absorbing sheet . . . 511
42.3.2. Perfectly Matched Layer . . . 511

43. Stability Conditions **512**
43.1. Uniform grid . . . 512
43.2. Non-uniform grid . . . 512
43.3. Open boundaries . . . 512

44. Excitations **514**
44.1. Types of excitation . . . 514
44.1.1. Concentrated source excitation . . . 514
44.1.2. Matched source excitation . . . 514
44.1.3. Plane wave excitation . . . 514
44.1.4. Waveguide excitation . . . 515
44.2. Time signals . . . 516

45. Ports and S–matrices **517**
45.1. Port definition . . . 517
45.1.1. Quasi-TEM ports . . . 518
45.1.2. Lumped ports . . . 519
45.1.3. Waveguide ports . . . 519
45.2. Time signals . . . 520
45.2.1. Absorbing Port signal processing . . . 521
45.2.2. Concentrated Port signal processing . . . 521
45.3. DFT . . . 521
45.4. Impedances . . . 522
45.5. Wave quantities . . . 522
45.6. Scattering parameters . . . 522

46. Specials **523**
46.1. Y-Parameter Processing . . . 523
46.2. Hollow Waveguides . . . 523

46.3. Near to Far Field Transformation 524
46.4. SAR and ACD 525
46.5. Low-Frequency Algorithm 525
46.6. Dielectric Properties of Biological Tissue 526

VII. Literature **527**

Bibliography **528**

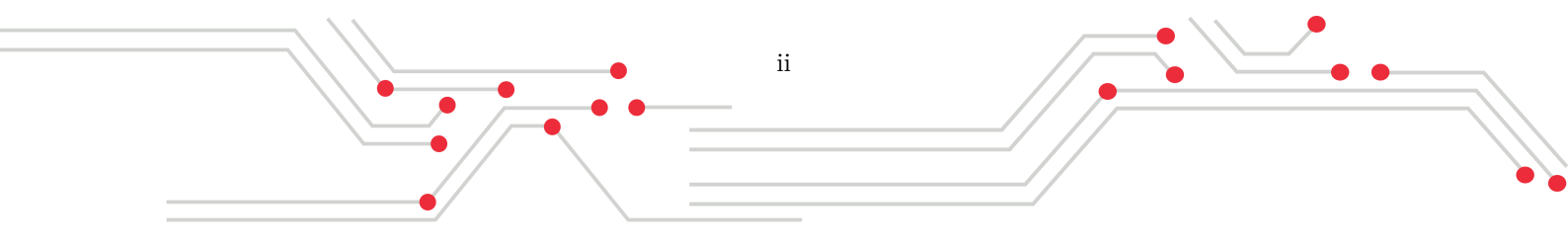
VIII. Index **529**

Reference index **530**
List of Actions 530
List of Options 532
List of Autodisc Options 533
List of Simulation Options 533

User and Reference Manual

for the
3D-EM Time Domain Simulator
EMPIRE XPU™
Version 9.2

©IMST GmbH
Carl-Friedrich-Gauß-Str. 2-4 • D-47475 Kamp-Lintfort, Germany
Phone +49 2842 981 0 • Fax +49 2842 981 199
E-Mail: empire.support@imst.de • Internet: www.empire.de



Copyright notice

Copyright ©1998-2026 IMST GmbH. All rights reserved. No part of the EMPIRE XPU™ software, that *needs* a license, may be used without a valid license from the IMST.

THERE IS NO WARRANTY, TO THE EXTENT PERMITTED BY APPLICABLE LAW. EXCEPT WHEN OTHERWISE STATED IN WRITING THE COPYRIGHT HOLDERS AND/OR OTHER PARTIES PROVIDE THE PROGRAM ”AS-IS” WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

THE ENTIRE RISK AS TO THE QUALITY AND PERFORMANCE OF THE PROGRAM IS WITH YOU. SHOULD THE PROGRAM PROVE DEFECTIVE, YOU ASSUME THE COST OF ALL NECESSARY SERVICING, REPAIR OR CORRECTION.

Please pay attention to the following facts

- EMPIRE XPU™ is a trademark of IMST GmbH.
- Portions of this software are owned by Spatial Corp. © 1986 – 2026. All Rights Reserved.

Preface

EMPIRE XPU™ is an established and versatile electromagnetic field simulator based on the Finite-Difference Time-Domain Method (FDTD). At IMST GmbH, this method has been applied and highly optimized to build a very efficient and extremely fast **E**lectro **M**agnetic field simulator for the analysis of **P**ackages, **I**nterconnects, **R**adiators, waveguide **E**lements and EMC problems.
The exact knowledge of electromagnetic fields and the propagation of waves is a prerequisite for the design of radio frequency (RF) elements. In former times, RF design tasks were based on measurement and simple models, leading to time and costs consuming design procedures. Today, affordable computer hard- and software have established and simulation programs can accurately predict the electromagnetic behaviour of new products. So, field simulators have become a new standard for RF calculations.
Unlike the Finite Element Method, which has long been established in computing electromagnetic fields, the FDTD method had its breakthrough in the late 80's when extensive research and development lead to a number of improvements in applying advanced boundary conditions, e.g. free space or waveguides, and, therefore, reducing significantly the area of simulation. Today, its applicability covers the whole area of three-dimensional (3D) field simulations for RF designer.
Originally intended to analyse planar passive structures, the EMPIRE XPU™ simulator was continuously extended to capture more and more high frequency applications. It has proven its functionality in numerous comparisons with measurements and was successfully applied in many different microwave component designs. The method is based on the most efficient algorithm known which is very accurate and simplification is hardly necessary since it solves the discretised Maxwell's equations directly.
The EMPIRE XPU™ Graphical User Interface works on Windows and Linux platforms which support OpenGL graphics. Structure data can be easily exchanged with other CAD programs for both 2D layout data (DXF 12, Gerber, GDSII, ...) as well as 3D data (ODB++, STL, ...). Multiple port scattering parameters, radiation patterns, field plots etc. are generated for a user defined frequency range within only one simulation process. Monitoring and animation can give physical insight into the electromagnetic wave phenomena while accurate results are obtained with little effort.

EMPIRE XPU™ features advanced field solver operations, like expert-level automatic meshing, parameter sweep, batch processing, remote control, optimization and cluster computing which allows a rapid design and analysis of RF components, antennas and circuits. Due to its unique on-the-fly compilation and multiple time-stepping approach EMPIRE XPU™ is able to extensively exploit modern processor architectures which leads to a performance of up to 150 billion cells per seconds (GCells/s) on a single machine[1] with the possibility to access the complete built-in RAM [1]. This performance allows the treatment of large-scale structures such as BGA packages, slotted waveguide antennas, automotive antennas, human body exposure, RCS calculation or large antenna arrays in reasonable time.

[1] measured on an AMD Dual Epyc 9755 workstation

About this manual

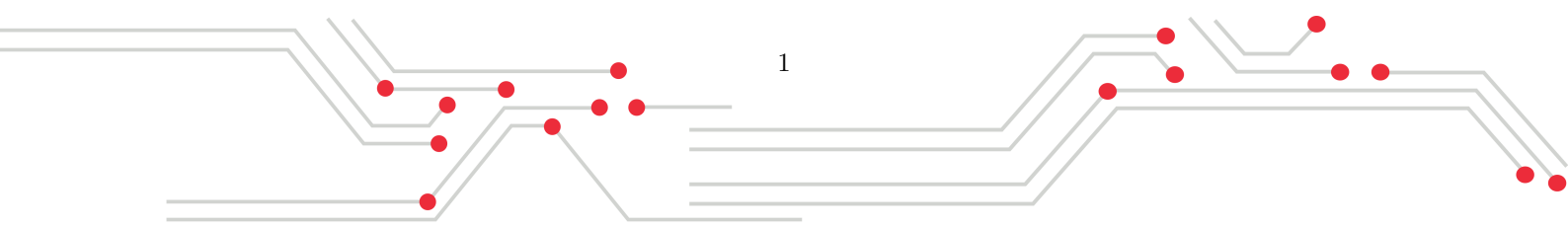
This manual is divided into several parts, intended to guide both very beginners and skilled experts from installation and simulation setup to understanding and exploiting the results.

- **P**art I describes the installation and licensing process for the different platforms.
- **P**art II explains the EMPIRE XPU™ Graphical User Interface, the simulation control as well as pre- and post-processing features.
- **P**art III is a quick guide for typical applications with references to the details given in the later chapters and sections.
- **P**art IV is the command and setup reference, a complete description of all actions, operations, options and simulation parameters of EMPIRE XPU™ .
- **P**art V gives a summary of warning and error messages along with comments to solve the problems.
- **P**art VI describes the theoretical background of the applied method.
- **P**art VIII is a keyword index with hyperlinks for a quick access to the designated topics in chapter and sections.

Any comments and suggestions related to this manual are highly appreciated and should be mailed to `empire.support@imst.de`.

Part I.

Software Administration



1. Installation on Windows PC

1.1. Requirements for PC Installation

The requirements are

- PC-Compatible (e.g. Intel Core i7/i9, Xeon, AMD Ryzen, Threadripper, Epyc) with at least 4 GB of RAM
- Microsoft Windows 10/11 or Windows Server (64 bit)
- Graphic resolution set to Full-HD (1920*1080px) or more
- Graphic processor with OpenGL support, incl. latest driver (e.g. Nvidia GeForce GTX 1060, AMD Radeon RX 480)
- 8 GB free hard disk storage

1.2. Installation Procedure

The user must have Administration rights to allow all files to be installed. It is recommended to execute all programs with **Right click - Run as Administrator**

Step 1:

- Insert the EMPIRE XPU™ media into your PC or unzip the downloaded file
- Run setup.exe:
  - Start the Windows Explorer
  - Change the directory to the EMPIRE XPU™ distribution
  - Start the program **setup.exe**
  - On request, allow permission to install the program
- The window shown in Fig. 1.1 will appear on your screen:

Step 2:

- Press `Next>` to continue the installation

Step 3:

- Please read the **License Policy**
- Press **I Agree** to continue the installation

Step 4:

- We recommend to install the full EMPIRE XPU™ installation
- If you want to install only the minimum configuration, deselect the corresponding check boxes
- Press `Next>` to continue the installation



Figure 1.1.: EMPIRE XPU™ Setup Start Window

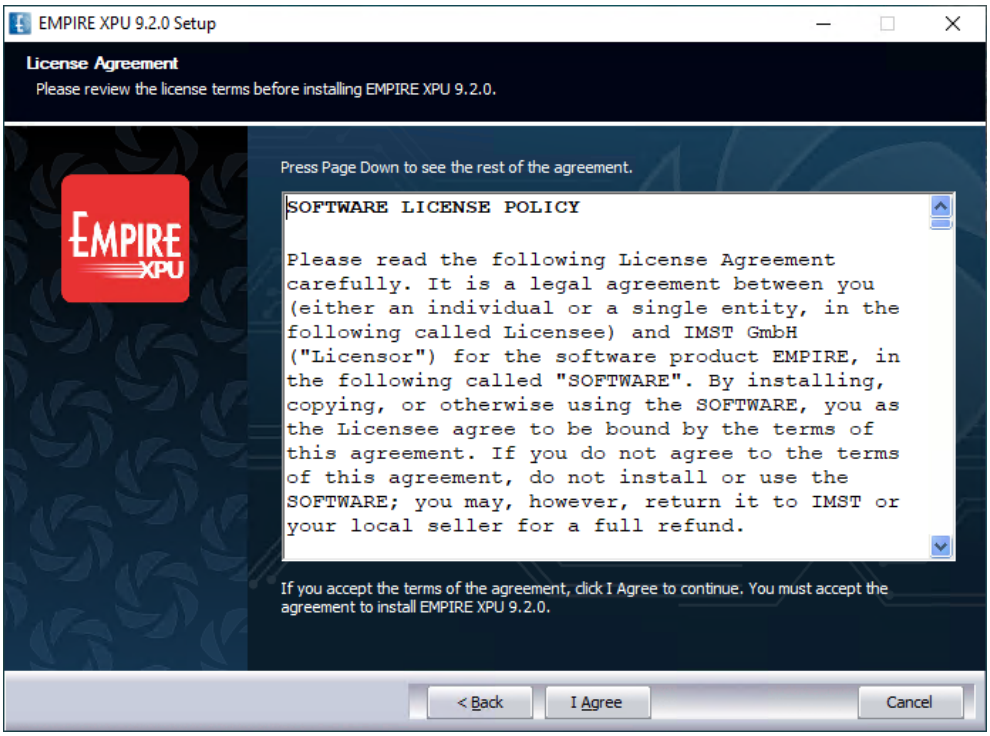


Figure 1.2.: EMPIRE XPU™ License Agreement Window

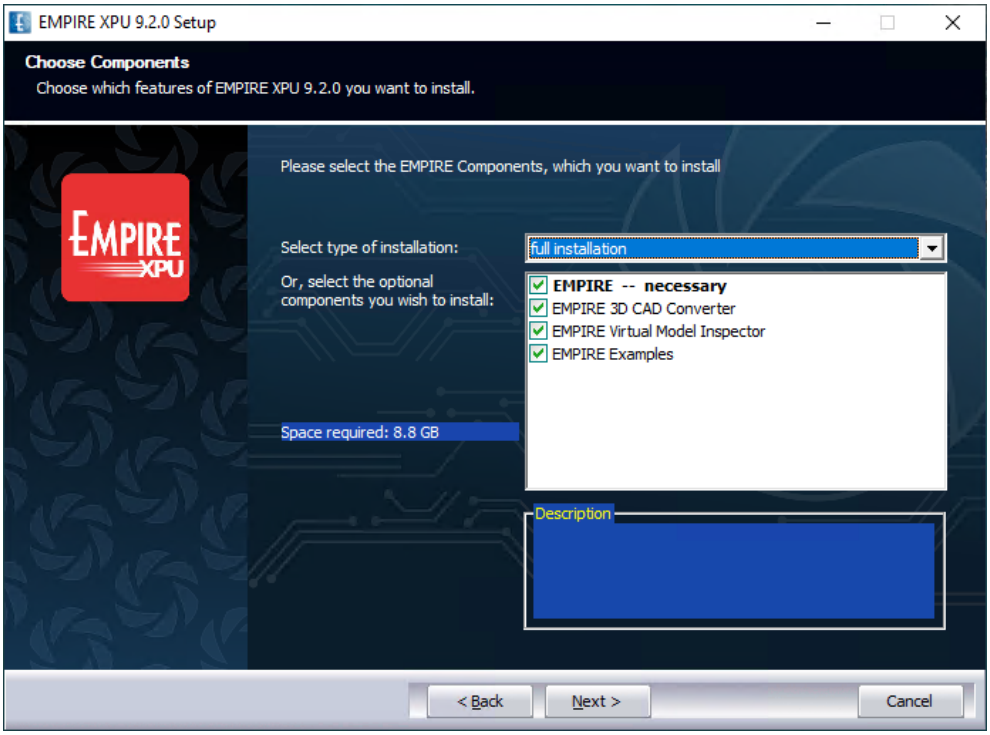


Figure 1.3.: EMPIRE XPU™ Choose Components Window

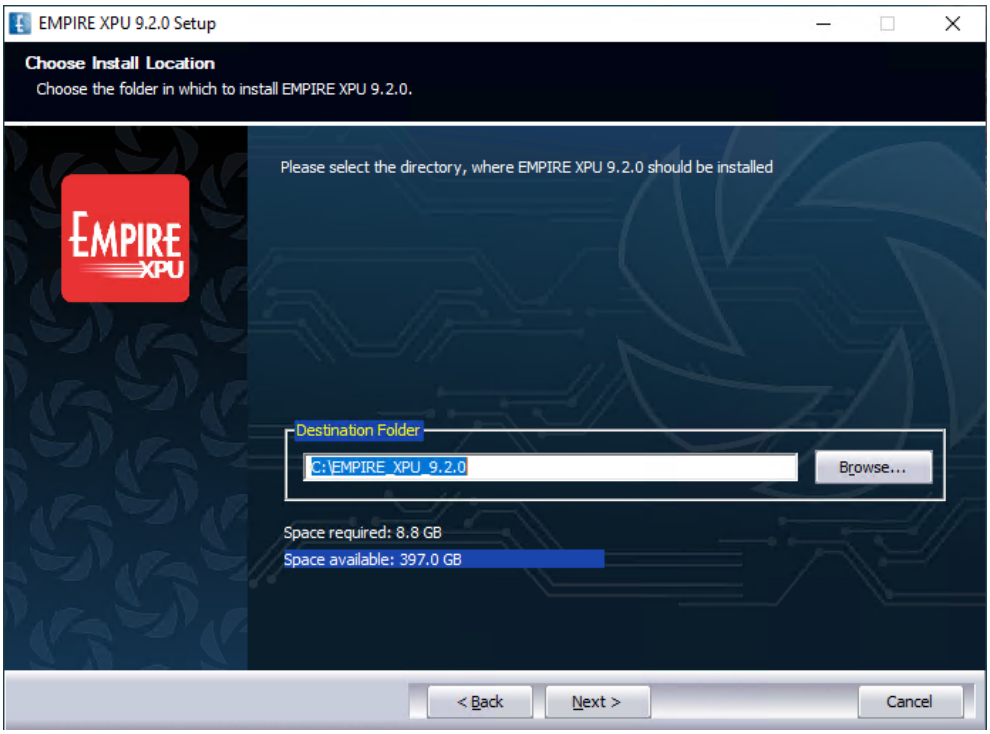


Figure 1.4.: EMPIRE XPU™ Choose Install Location Window

Step 5:

- Please confirm or browse the **Destination Folder**[a]. Do **not** use
  - folders with spaces such as **Program Files**[b]
  - folders containing special or non-UTF-8 characters
  - network drives
- Check the space required and if your system has enough memory available
- Press `Next>` to continue the installation

[a] The user should have write and execute permissions in this folder and it should be unblocked by anti-virus programs to allow license check-out

[b] On some systems spaces are not allowed in installation path.

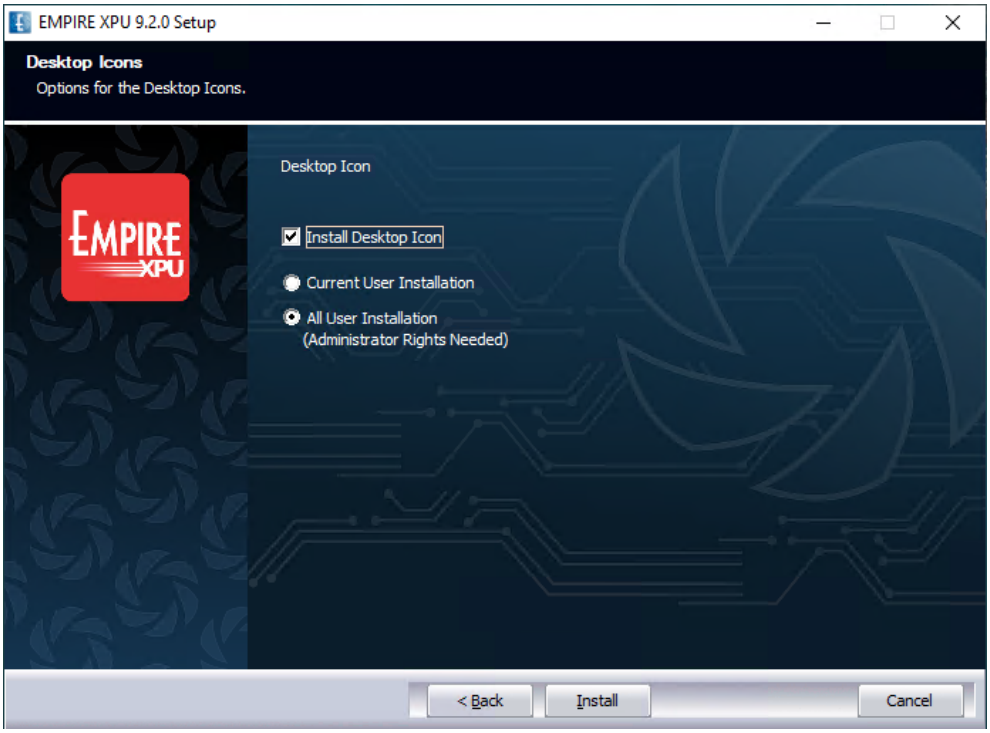


Figure 1.5.: EMPIRE XPU™ Install Desktop Icon Window

Step 6:

- Deselect the check box, if you do not want to install a **Desktop Icon** for EMPIRE XPU™
- Choose **Current User** or **All User** Installation. In case of **All User** installation the desktop icon will be available in `C:/Users/Public/Desktop/` and is available for all users.
- Press `Install` to continue the installation

Step 7:

- The installation of the EMPIRE XPU™ files is ready when the `"Next >"` button is available
- Please proceed by pressing `Next>`
- Close the Setup Window by pressing **Finish**
- You can now start EMPIRE XPU™ with the **Desktop Icon** or from the **Start Menu**
- You will need a license for running EMPIRE XPU™, please continue at Chapter 2 for the license installation

Hint

If the EMPIRE XPU™ Desktop icon is not available or double-clicking the icon does not open any window please repeat the installation process again with another installation path with no spaces or special characters, e.g. `C:\Empire_XPU`

1.3. Multiple version installations

It is possible to install multiple versions of Empire provided separate installation folders are used and desktop icons are named uniquely.

The file association (e.g. double clicking an input file) is always done with the latest installed version.

Since version 7.6 a function called **Set file association** can be found in the **Utilities** menu.

1.4. Silent installation on Windows

A silent installation can be done with following procedure:

- Extract the distribution and locate the folder containing **setup.exe**
- Create and run a **.bat** file or use a **cmd** window and run the command in the distribution folder: `.\setup.exe /S /NCRC /D=C:\EMPIRE_XPU_9.2.0\`
- Optionally, use another installation folder after `/D=`

Note, that no Desktop icon is created with this method. If required, drag it from the Start Menu to the Desktop.

1.5. Uninstall

The de-installation can be done with several procedures:

- **EMPIRE XPU - Uninstall** entry in the Windows Start menu
- Executable **uninstall.exe** which is located in the installation folder
- Use the Windows System program deinstallation feature

The uninstall cleans up the installation folder, keeping possible license data, only.

2. License Setup

Without a license the program can only be started for browsing existing projects or examples. Saving and running projects require a valid license.

The license can be node locked to a designated PC, dongle, or floating in a network. In the latter case one computer has to act as the license server.

2.1. License identification numbers

By default, licenses will be locked to the **DISK_SERIAL_Number** on Windows and to the **Ethernet address** on Linux systems. On request the licenses can be locked to Ethernet addresses or hard-lock keys (Dongles) on Windows systems.

The id numbers can be obtianed with the following methods

- **License Request** Starting Empire and clicking the **Request** button, see Fig.2.3
- **License Management Tool** Starting **lmtools.exe** in `C:\EMPIRE_XPU_9.x.x\empire\flexlm\winnt` switch to **System Settings** Tab, see Fig.2.1
- **Command Line**
  - **Disk Serial Number**: Command: `vol c:`
  - **Ethernet Address**: Command: `ipconfig /all`

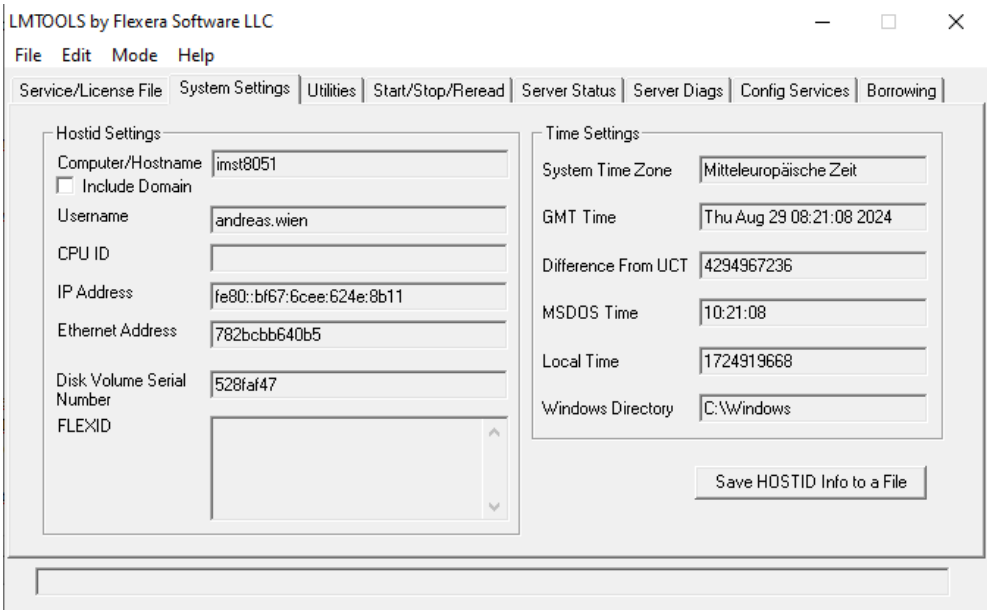


Figure 2.1.: License Management Tool

2.2. Overview of Licensing Options

1. **Node-Locked License** With this option, the license key is locked to the PC hardware. The license is not transferable to another PC and therefore EMPIRE XPU™ is limited to running on that computer, only. On a Windows based system, the license is locked to the hard drive or MAC identification number. On a Linux based system, the license is always locked to the MAC address.
2. **USB Dongle** With this option, the license key is locked to a hard-lock dongle. Therefore, EMPIRE XPU™ can run on any PC where the dongle is plugged into (and where the EMPIRE XPU™ software is installed). This only works with Windows based PCs. A driver has to be installed which can be found in the installation folder.
3. **Floating License** With this option, a license service runs on a designated server (or even a triad for redundancy reasons). Concurrent licenses can be obtained for any client PCs from the license server through the network. This option works with both Windows and Linux based PCs. For example, the server operating system could be Linux (or Windows), and the client PC operating systems could be mix of Windows and Linux.

In case of purchase, the license keys (for Node-Locked, Dongle and Floating) are renewed on a yearly base by default. The new license keys are emailed to customers prior to the expiration date of the old license. This service enables the user to exchange his hardware from time to time. On request the license can be issued without expiration date, but the hardware can not be changed without additional costs.

In case of renting the license keys are issued for the renting period.

2.3. Dongle licensing

EMPIRE XPU™ supports the following dongles on Windows:

- **Aladdin**, USB dongle, FLEXid=9

Dongles can be obtained from IMST for a shipping and handling fee.

Before the dongles can be recognized by a Windows PC a driver has to be installed once.

1. If not already installed, run setup.exe from EMPIRE distribution (e.g. download, USB-Stick) and confirm messages
2. Install dongle driver
   a) Run `FLEXId_Dongle_Driver_Installer.exe`, available in installation folder, e.g. `C:\EMPIRE_XPU_9.x.x\empire\flexlm\winnt`
   b) In Select Options, Select **FLEXid 9 Drivers**, only, deselect other FLEXid driver numbers
3. Attach Flexera Dongle to your system

After restart and if the installation is successful the Dongle FLEXid number will be displayed in the EMPIRE XPU™ license request file.

Hint: Dongle driver on Windows 10/11

The current driver may report incompatibility with Windows 10 for FLEXid numbers 6,7, and 8. This warning can be disregarded. The installation will be carried out nevertheless.

In order to remove old or defective dongle drivers the tool FLEXidCleanUtility.exe is supplied in the mentioned folder, too.

2.4. Request a License

To get the license for the EMPIRE XPU™ software some identification numbers of your computer on which the software shall be installed are required to generate the key.

Step 1:
- Start EMPIRE XPU™, e.g. from Desktop

The start-up screen will appear as shown in Fig. 2.2

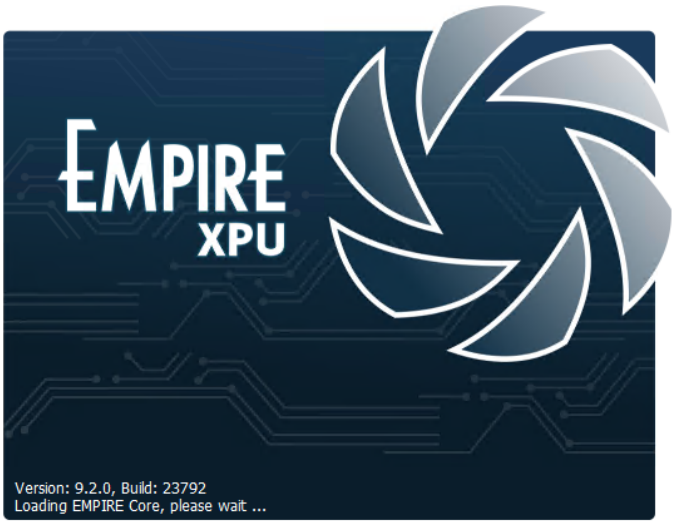


Figure 2.2.: EMPIRE XPU™ Start-up screen

If no license can be found or a license has expired the following window as in Fig. 2.3 will come up.

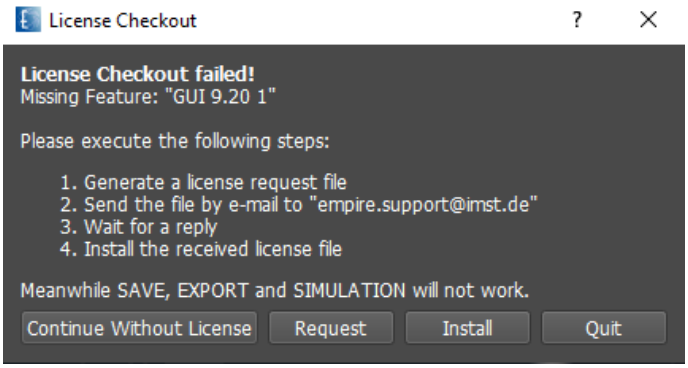


Figure 2.3.: EMPIRE XPU™ License checkout failure

Step 2:
- Click **Request**

Step 3: The **License Request Form** will open as in Fig. 2.4
- Please complete the form with E-mail if different to your sending address
- Save the text into a file

Step 4: Send an E-mail to **empire.support@imst.de** and attach the saved file.

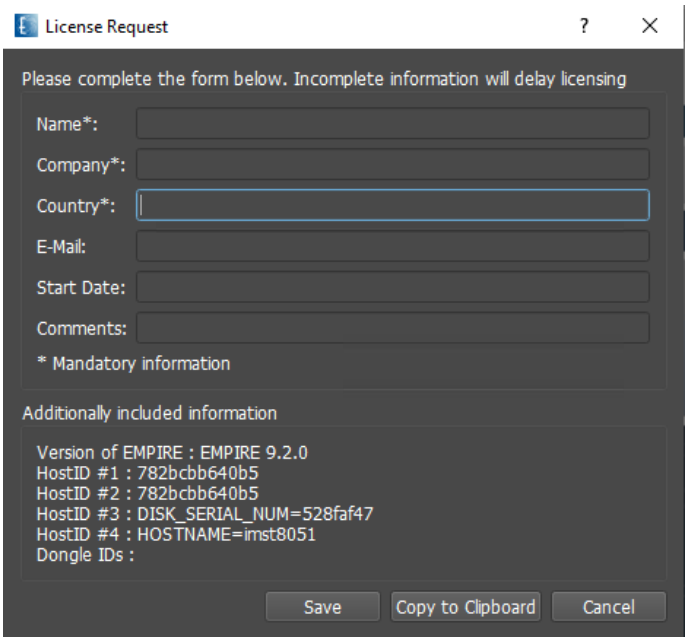


Figure 2.4.: EMPIRE XPU™ License Request Window

Hint

The license request form can also be opened with the menu item **File - License - License request**.

2.5. Install the License

You need write permission to the EMPIRE XPU™ installation folder to install the license keys.

The EMPIRE XPU™ Support Team will generate a license file and send it to you.

Step 1:

- Save the license file that you received by E-mail

Step 2:

- Start EMPIRE XPU™:
  - On Windows 10/11 click right mouse button on icon and start it with Admin rights
  - On Linux enter EMPIRE-XPU in the command line
- Click **Install**
- Click **Accept** if you agree to the licensing terms
- The license installation is completed
- Please close EMPIRE XPU™ and start it again.

Hint: Manual License installation

The license installation is a copy and rename action and can be done manually:

- Save the license file into the following location
  `"C:\EMPIRE_XPU_9.x.x\empire\flexlm"`
- Rename the saved file to `license.dat`

See chapter 4.4 License diagnostics in case of licensing problems.

2.6. License File

The following features are available and licensed depending on the purchased package:

emtwin	EM-TWIN module
empire-fs	EM solver, FDTD kernel
ganymed	structure editor
empire-rc	remote computing
empire-nf	far field transformation
empire-hw	TE/TM waveguide modes
empire-cd	human body SAR ACD EIAV averaging
empire-lf	human body low frequency algorithm
empire-ve	human body voxel editor
empire-op	optimizer
empire-mpc	parallel simulation, multi-PC solver
empire-th	thermal solver
empire-px	human body voxel poser
empire-cs	circuit simulation
empire-vmi	virtual model inspection module
empire-odb	ODB++ import
empire-acs-im	ACIS import
empire-igs-im	IGES import
empire-stp-im	STEP import
empire-nxd-im	NXD import
empire-dxf-im	3D DXF import
empire-inv-im	Inventor import
empire-par-im	Parasolid import
empire-pro-im	ProE import
empire-swd-im	Solid Works import
empire-sed-im	Solid Edge import
empire-jtd-im	JT import
empire-ct4-im	CATIA 4 import
empire-ct5-im	CATIA 5 import
empire-acs-ex	ACIS export
empire-igs-ex	IGES export
empire-stp-ex	STEP export
empire-ct4-ex	CATIA 4 export
empire-ct5-ex	CATIA 5 export

The license file is a plain-text file which contains license keys for different features. Each feature line contains the vendor daemon name, version number, expiration date, ID numbers, keys and signatures. In case of node-locked licenses the number of licenses is uncounted, e.g. multiple editors may be opened at the same time. In case of floating licenses the number of licenses is written, as well as hostname, port and vendor daemon.

Hint

The (server) hostname and port number can be edited in case of floating licenses

3. Starting Empire on Windows

After installation and license setup, EMPIRE XPU™ can be started on Windows with

- Double click the desktop icon **EMPIRE XPU 9.x.x**
- Double click an existing file with the ending `.emx` (or `.gym` for compatibility reasons)
- Start Menu - EMPIRE XPU™ 9.x.x - empire-9.x.x
- Double click **EMPIRE.cmd** in `C:\EMPIRE_XPU_9.2.0\empire\binW`
- a command entered in the command window (no line break):
  `start "" "C:\EMPIRE_XPU_9.2.0\empire\PY3k-W64\Python3k\python.exe" -O -u "C:\EMPIRE_XPU_9.2.0\empire\env.py" kallisto`

All necessary data for a simulation, like geometry, electric properties, simulation parameters, mesh, are stored in an input file ending with `.emx`. A sub folder is automatically created where results and intermediate files will be stored.

Before simulation, an intermediate file format `.acad` is generated for the simulation kernel. This is done by pressing the buttons **Start Simulation** or **Preprocessing** in the **Simulation** tab.

Hint: Conflict with other programs

If another Python-based program is installed (e.g. MatLab) there may be a conflict when starting the simulation and a wrong Python version is used. This can be avoided by defining the **Path** variable before starting the command above, see section 3.2.

3.1. Simple Test Procedure

After software and license installation the EMPIRE XPU™ program can be tested quickly with the following procedure:

3.1.1. License test

1. Double click desktop icon
   - Check: if the full screen comes up, see Figure 3.1, the license is working

3.1.2. Graphic test

2. Select the **Templates** tab and click OK.
   - Check: If the MSL line example is visible, see Figure 3.2, the OpenGL graphic is working

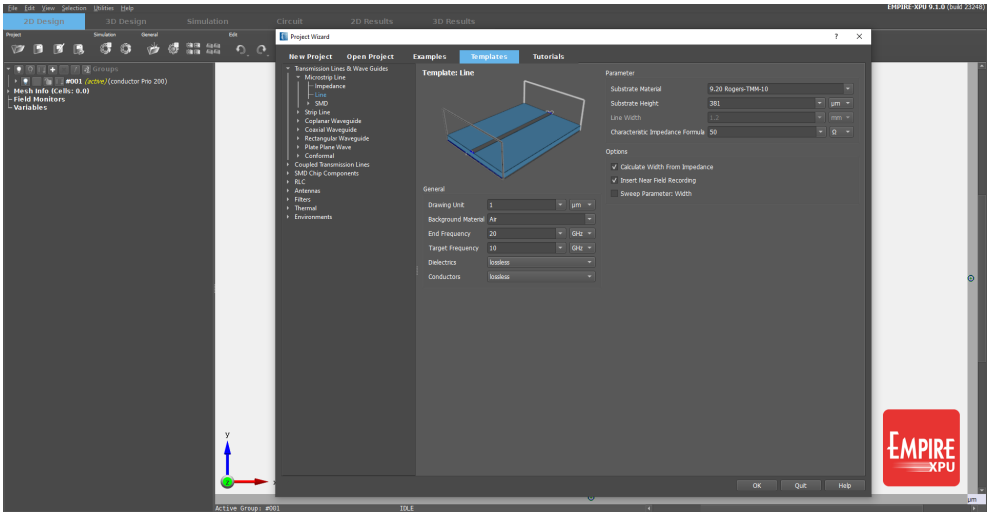


Figure 3.1.: EMPIRE XPU™ Start-up Window

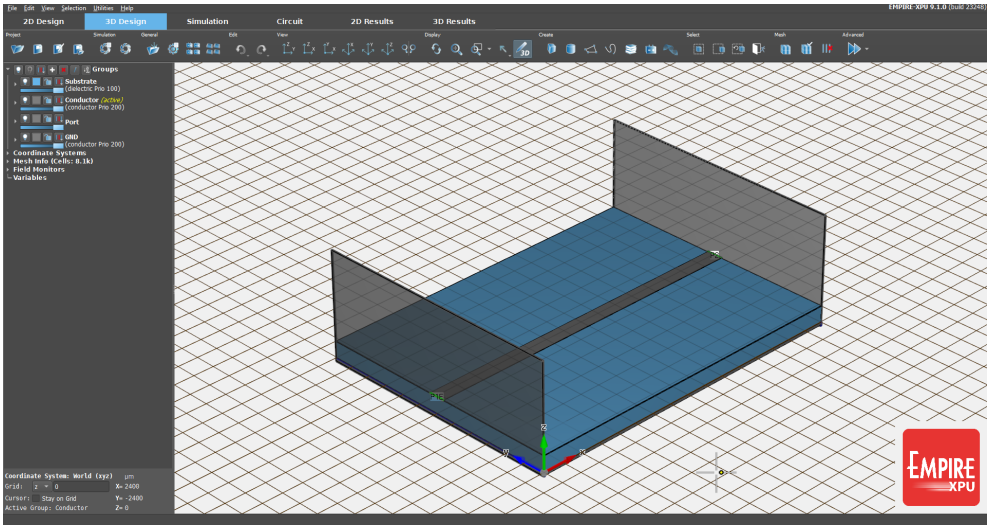


Figure 3.2.: EMPIRE XPU™ Standard Template

3.1.3. Compiler test

3. Click Button "Start Simulation",  confirm messages
   - Check: If the **Simulation** Tab shows the energy convergency plot over timesteps at the end, see Figure 3.3, the compiler and simulation kernel are working

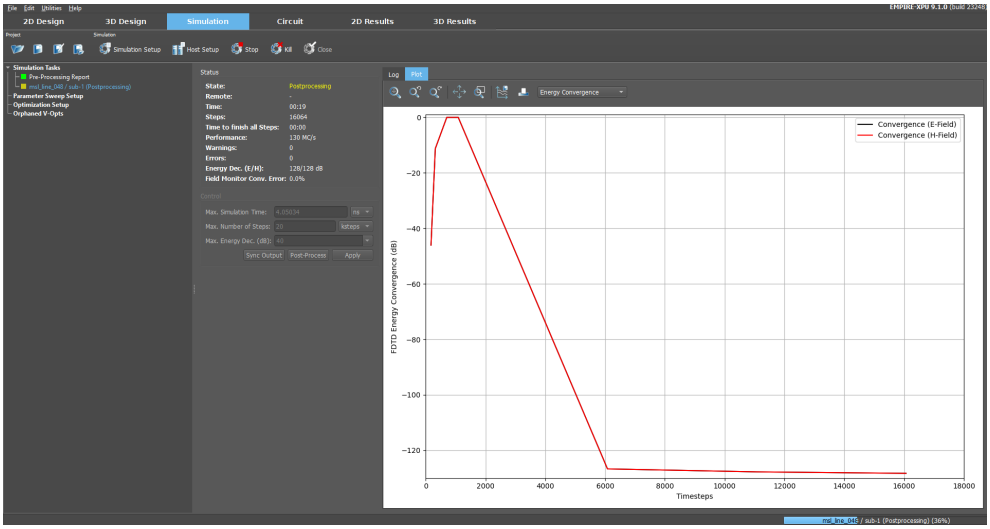


Figure 3.3.: EMPIRE XPU™ Standard Template simulation convergence

3.1.4. Plot library test

4. Click Tab "2D Results",
   - Check: If the S-Parameter results are plotted as in Figure 3.4 the simulation was successful and the plot library is working

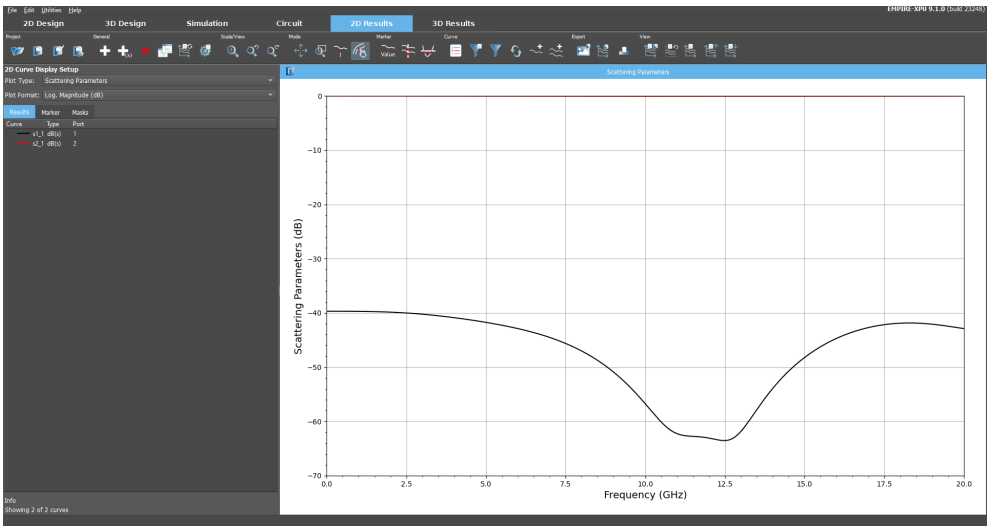


Figure 3.4.: EMPIRE XPU™ 2D Result Plot

3.1.5. Visualization tool kit test

5. Click Tab "3D Results", switch on light bulb in **Field Monitors**, switch off light bulb in group **Substrate**, optionally adjust view
   - Check: If the field is displayed, see Figure 3.5, the Vizualization Tool Kit (VTK) works

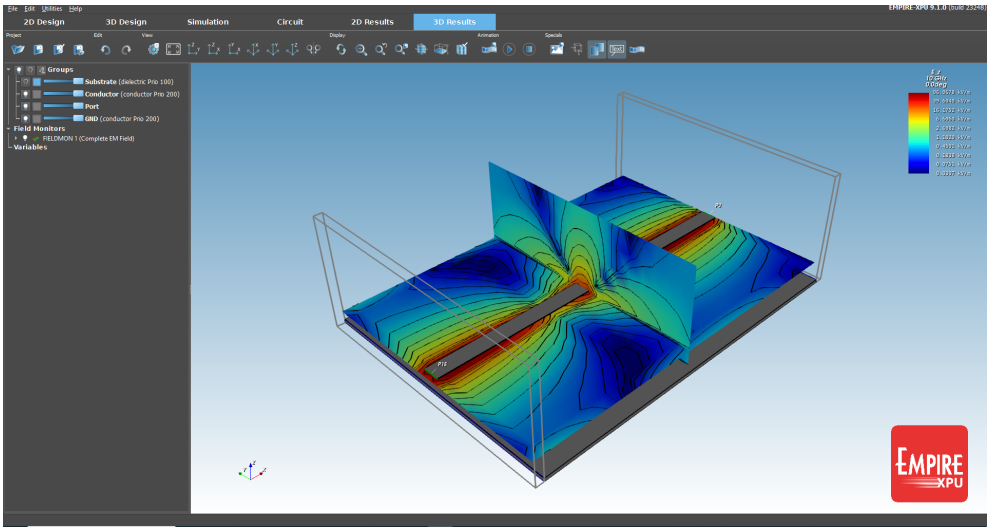


Figure 3.5.: EMPIRE XPU™ 3D Result Plot

3.2. Simulation from the Windows command line

It is also possible to preprocess the structure and run a simulation from a command line using the following command (no line break, replace x.x by actual version number):

```
"C:\EMPIRE_XPU_9.x.x\empire\PY3k-W64\Python3k\python.exe" -O -u
"C:\EMPIRE_XPU_9.x.x\empire\env.py" simul msl_line.emx
```

Remarks

- It is assumed that the command is executed in a folder which has an input file, ending **.emx**
- The first part of the command starts the program **python.exe** with absolute path and some options
- The second part runs a script **env.py** with absolute path
- If executed with option: `kallisto msl_line.emx export`, the command converts an input file **.emx** and creates an intermediate file **.acad**
- If executed with option: `proc msl_line\msl_line.acad`, the command starts a simulation with a given **.acad** file in the current project folder

If another Python-based program is installed (e.g. MatLab) there may be a conflict when starting the simulation and a wrong Python version is used. This can be avoided by defining the **Path** variable before starting the command above:

```
set PATH=C:\EMPIRE_XPU_9.x.x\empire\PY3k-W64\Python3k
```

3.3. Graphic Renderer Information

The software makes use of advanced OpenGL graphic features. The used graphic card and driver can be checked with:

- Select the **3D Design** Tab
- Select **Help - Info - System Info** Tab
- Check if the best available graphic card is selected. Optionally activate the graphic card in its system control
- Check if the latest driver is used for this graphic card. Optionally download the latest driver from the card vendor and install it

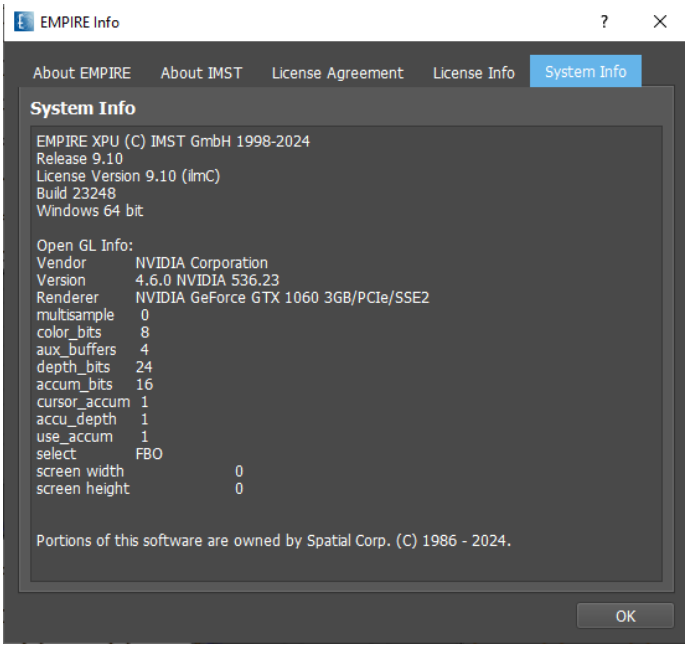


Figure 3.6.: EMPIRE XPU™ System Info

4. Floating License on Windows

This chapter describes the setup of floating licenses. With this option, a designated PC acts as a **license server** from which all clients in the network can check out available licenses. It is not necessary to install the complete EMPIRE XPU™ on this server, the following programs which can be found in the EMPIRE XPU™ installation `...\empire\flexlm\winnt` are sufficient:

- **lmtools.exe** Program to start and stop the service, license diagnostic
- **lmgrd.exe** License Manager Daemon
- **imstlmd.exe** Vendor Daemon

The daemons use two ports:

- One is a port for the "lmgrd.exe" daemon (by default 1700)
- The other is a port for the "imstlmd.exe" daemon. By default it is selected at runtime and displayed in the log file of the Flexlm Service

Hint: License Server hostname

The hostname (e.g. mylicserv) should be known and reachable from the client, e.g. the command `ping mylicserv` succeeds on the client. If not, it is also possible to use the IP number instead of the hostname.

If needed the hostname and port numbers can be changed in the license file.

4.1. Floating License File

In addition to the entries of the node-locked license file the floating license file contains a server and a vendor daemon line.

Standard	Modification example

`SERVER hostname id# 1700`
`DAEMON imstlmd`
`FEATURE empire-fs imstlmd ver# date lic# ...`
`...`
	`SERVER mylicserv id# 55005`
	`DAEMON imstlmd port=55006`
	`FEATURE empire-fs imstlmd ver# date lic# ...`
	`...`

The server line contains the hostname, ID number and port number. Both hostname and port number can be edited by the end user. If needed the port number of the vendor daemon may be specified, too. The version number, expiration date and number of licenses are limited and written in the feature lines.

4.2. Installation on the Server

For the floating license, additional steps have to be taken to ensure that the license daemon is started automatically on the license server at power up:

In the following lines, it is assumed that the daemons **lmgrd.exe**, **imstlmd.exe** and **lmtools.exe** are available in a folder called `A\empire\flexlm\winnt` (A is `C:\EMPIRE_XPU_9.x.x` in a standard installation)

- If EMPIRE XPU™ is installed completely on the server:
  - Install the license as described in section 2.5
- If only daemons are installed on the server:
  - Save the license file as `license.dat` in folder `A\empire\flexlm`
- Use the Windows Explorer and change to the folder `A\empire\flexlm\winnt`
- Start the program "lmtools.exe" (right mouse click, start with Admin rights)
- Choose the following settings:
  - Select radio box **Configuration using Services** as in Fig. 4.1
  - Select tab **Config Services** as in Fig. 4.2
  - Enter a service name i.e. "Empire License Service"
  - Set the paths
    - `A\empire\flexlm\winnt\lmgrd.exe`
    - `A\empire\flexlm\license.dat`
    - Optionally, specify a debug file `A\empire\flexlm\debug.log`
  - Select: "Use Services" and "Start Server at Power up"
  - Press "Save Service"
- Please make sure that the executable files

  `A\empire\flexlm\winnt\lmgrd.exe` and

  `A\empire\flexlm\winnt\imstlmd.exe`

  are not fire-walled.
- Leave the program and reboot the PC.

Hint: Manual floating license setup

The license server can be started manually if a license service has been defined as written above, e.g. to test the floating license

- Start the progam "lmtools.exe"
- Go to the tab **Start/Stop/Reread**
- Press **Start Server**

The tabs **Server Status** and **Server Diags** are helpful to detect problems with the license.

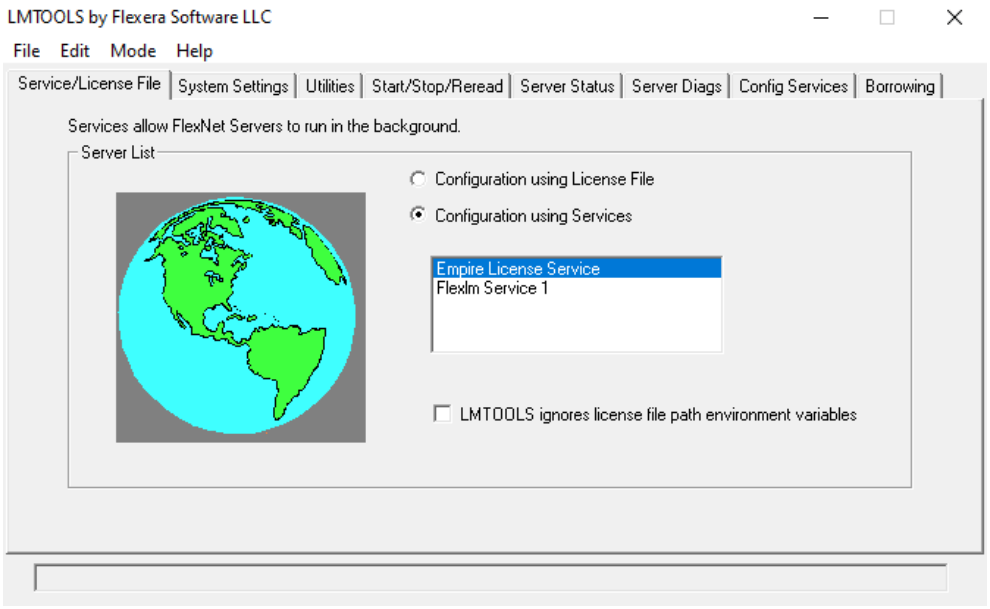


Figure 4.1.: EMPIRE XPU™ Lmtools Starting Window

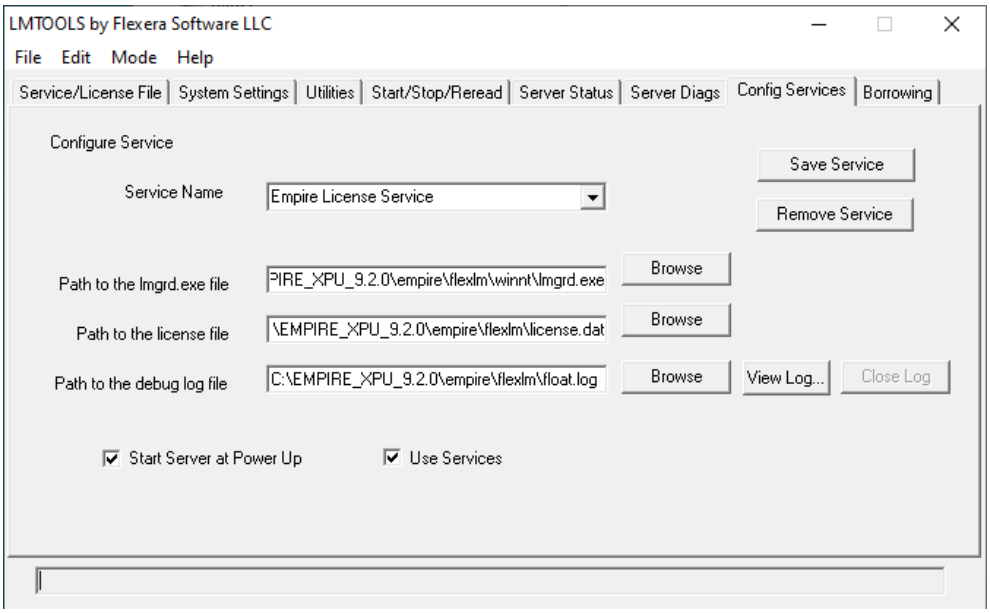


Figure 4.2.: EMPIRE XPU™ Configure Services Tab

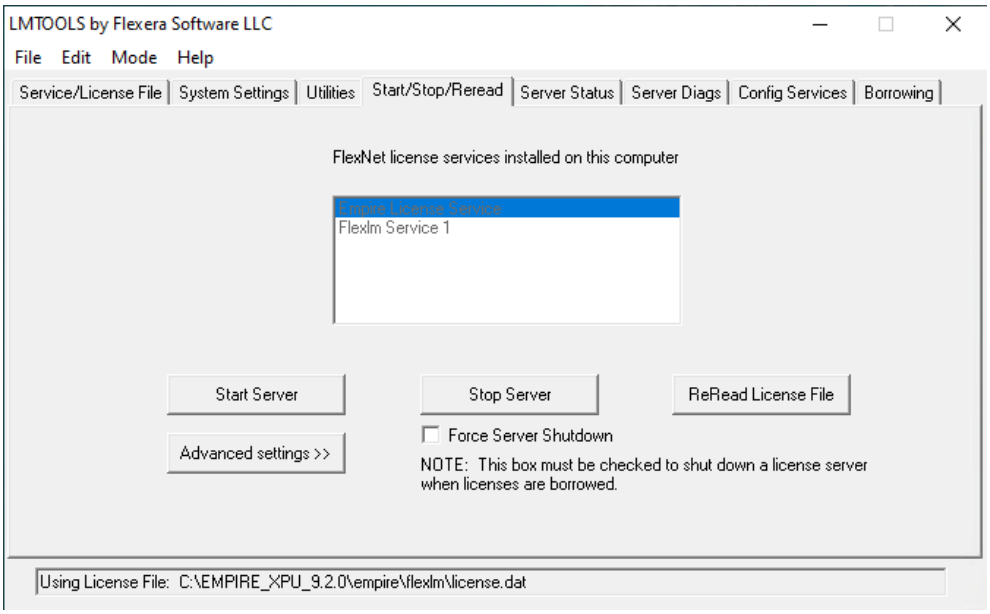


Figure 4.3.: EMPIRE XPU™ Lmtools Start Server Tab

4.2.1. Delayed license checkout

It has been reported that in some cases checked-out licenses are returned with a 2-hours delay after exiting the used feature. This can happen if the TCPIP connection of the licensing daemon is broken for some reason. In this case the system is waiting 2 hours before the license is released again.

There is a standard ”keep alive time” for broken TCPIP communications which is set to 2 hours. This period can be reduced to e.g. 2 minutes with the following actions:

On a Windows License server the following has to be set by the administrator:

1. Stop the license server
2. In the registry (cmd regedit)
   - Search the entry `HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Services\Tcpip\Parameters`
   - Create Parameter: KeepAliveTime, Type: REG_DWORD
   - Set Value: 120000 Decimal (= 2 Minutes)
3. Confirm and restart the server

4.3. Installation on each Client

Each client has to install the floating license file of the server as described in section 2.5. The firewall settings as mentioned before should be checked. EMPIRE XPU™ assumes the license with name **license.dat** to be located in a certain local folder: `C:\EMPIRE_XPU_9.x.x\empire\flexlm`

It recognizes the server hostname which is written in the license file and directs the request to the server.

Hint: Floating License on Client

It is essential that the license files on server and clients are the same for proper communication. E.g. after editing port number, the license file needs to be updated on all servers and clients.

As an alternative EMPIRE XPU™ can be started with an option `-nolm` in the target line of the Desktop icon. Then EMPIRE XPU™ uses the environment variable `LM_LICENSE_FILE=port@hostname`

4.4. License Diagnostic

A tool called `lmtools.exe` is available in the installation tree `C:\EMPIRE_XPU_9.x.x\empire\flexlm\winnt` which helps to detect license problems. By executing this program and selecting **Server diagnostics** the window as in Figure 4.4 appears. Here the status of the licenses can be checked. Further, by selecting **Server Status** (Figure 4.5) it can be checked if the server is running correctly. The server can be controlled manually in the **Start/Stop/Reread** section of this tool. If there are troubles concerning the license management please attach the output of the diagnostics to your email.

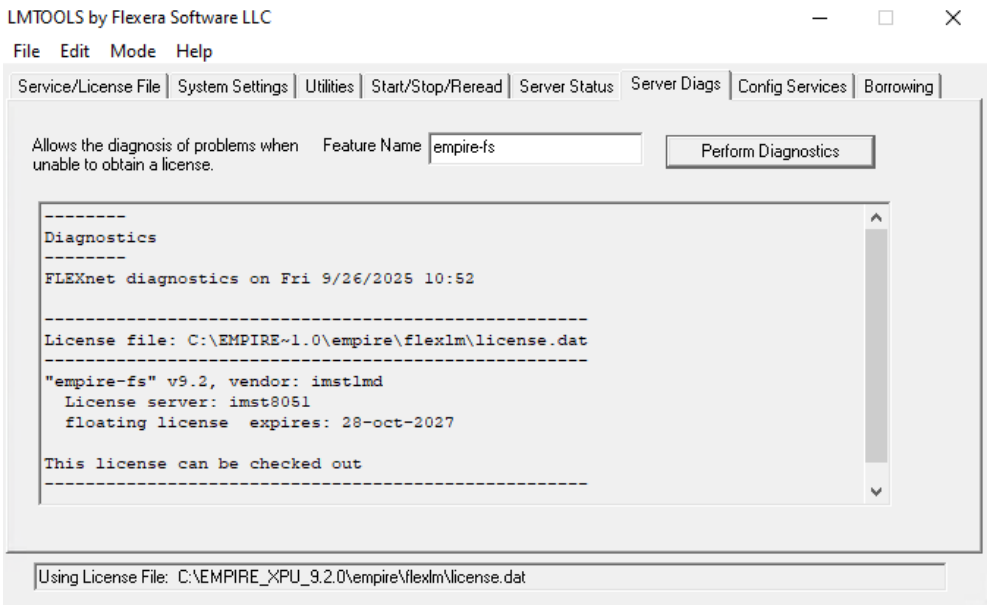


Figure 4.4.: EMPIRE XPU™ Lmtools Server Diagnostic

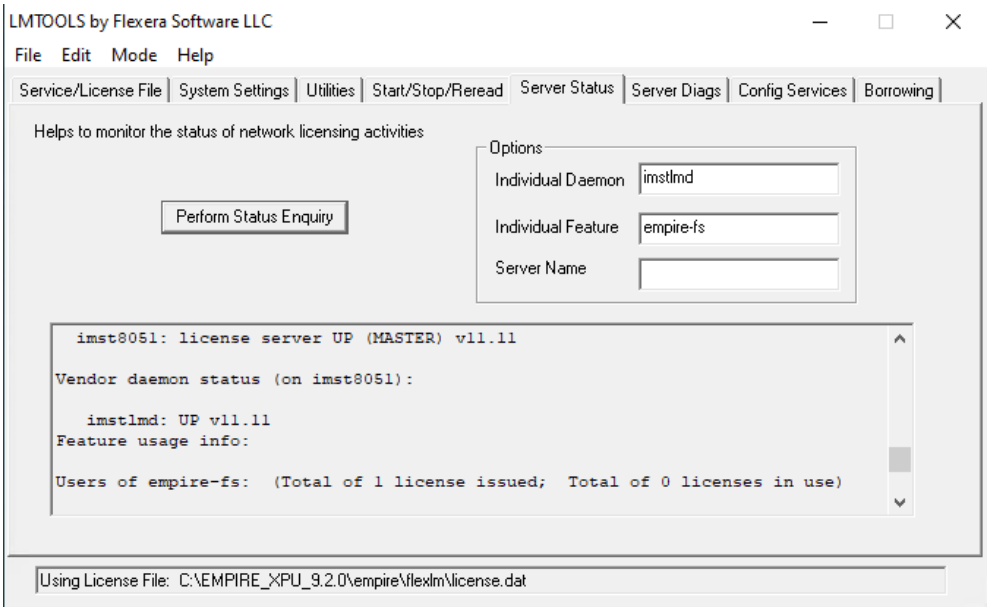


Figure 4.5.: EMPIRE XPU™ Lmtools Server Status

5. Remote Control

Basically, there are two options to work with EMPIRE XPU™ remotely. The GUI can be controlled on a Remote Desktop or the GUI is run on the local PC **Client** and the simulation kernel is started on a **Server** in a LAN.

5.1. Remote Desktop

5.1.1. Windows Remote desktop

In this mode the user has access to the network via LAN or VPN. The Windows Remote Desktop Connection program can be used to control the PC on which EMPIRE XPU™ is installed, see Figure 5.1

Hint

It is required that the Color is set to highest quality (32bit) in the Remote Desktop Connection Options.

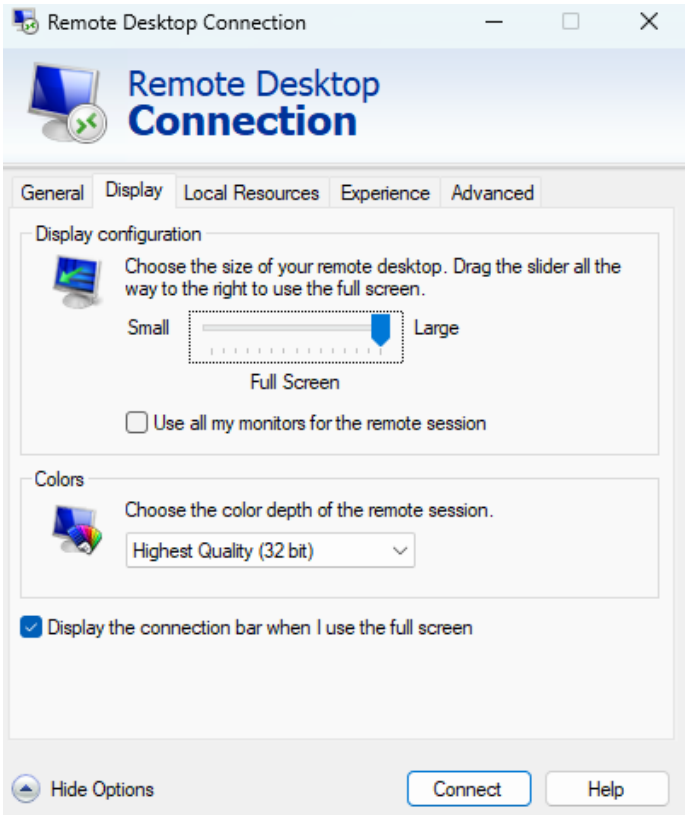


Figure 5.1.: Configuration for Windows Remote Desktop Connection

- For working in the **3D Design** mode, it is recommended to set the **Display Quality** in the **Editor Options** to a low value.
- Some graphic cards do not support the OpenGL commands used for the **3D Results** mode using remote desktop (e.g. Nvidia GeForce). If possible, select an on-board Intel graphic card for Remote Desktop connections or install a vendor update if available (e.g. **nvidiaopenglrdp.exe** supplied by Nvidia)

5.1.2. PCoIP

In order to obtain the best graphic quality a host remote workstation card can be recommended, e.g. **Tera Dici**. Here the graphic card of the remote server is connected to the PCoIP card. A small remote box or a software client can be used locally.

5.2. Remote computing

The EMPIRE XPU™ remote control allows to run simulations on other computers from the local computer in a LAN. The other computers (except for the slaves in multi-PC computing) need to have a license, which can be a Node Locked, Hard-locked, or Floating Network License.

In order to run simulations remotely an EMPIRE XPU™ Remote Service has to be started which runs in the background. There are two services:

- **Server** Reconnectable server for remote simulation. (Does not support multi-PC computing)
- **Legacy Server** Server for multi-PC computing and remote simulation (client-server connection may not be interrupted during simulaiton)

Start Service

- Service started within EMPIRE XPU™ in **Utilities - Server - Start** or **Utilities - Server (Legacy) - Start**
- Start associated Windows Service (Legacy Server)
- From Command Line:
  - Windows: `EMPIRE-SERVER.cmd` or `EMPIRE-SERVER-LEGACY.cmd` (located in `C:\EMPIRE_XPU_9.x.x\empire\binW`)
  - Linux: `./EMPIRE-SERVER` or `./EMPIRE-SERVER-LEGACY`

Then the EMPIRE XPU™ client can run the simulations on the remote server in Batch jobs, Parameter Sweeps, Optimizations or parallel on a cluster. The remote servers will be listed in the host lists which are available in **Host Setup** of the Simulaiton Tab.

The remote service requires that the hostnames are known in the network, e.g. resolved using a DNS server. If this is not available it is also possible to use the IP adresses which is described in section 5.2.5.

For remote processing in the network ports 42000 (TCP and UDP) and 43000 to 43500 (UDP) are used and should not be blocked by a firewall.

Hint: Server Selection

- The **Legacy Server** can be used for job distribution and job sharing (Mutli-PC algorithm). It requires a stable network connection and may not be disconnected during the simulation.
- The new **Server** can be used for job distribution, only. Client and server may be disconnected and reconnected during a simulation.

Manually Stop Service

- Service stopped within EMPIRE XPU™ in **Utilities** - **Server** - **Stop** or **Utilities** - **Server (Legacy)** - **Stop**
- Stop associated Windows Service
- From Command Line:
  - Windows: `EMPIRE-KILL-SERVER.cmd` or `EMPIRE-KILL-LEGACY-SERVER.cmd` (located in `C:\EMPIRE_XPU_9.x.x\empire\binW`)
  - Linux: `./EMPIRE-KILL-SERVER`

5.2.1. Starting the remote service in EMPIRE XPU™

Using this option, the remote service will be available until rebooting the remote server.

- Start EMPIRE XPU™ from the desktop of the remote server
- Select the menu **Utilities**
- Select **Server** - **Start** or **Server (Legacy)** - **Start**, see Figure 5.2

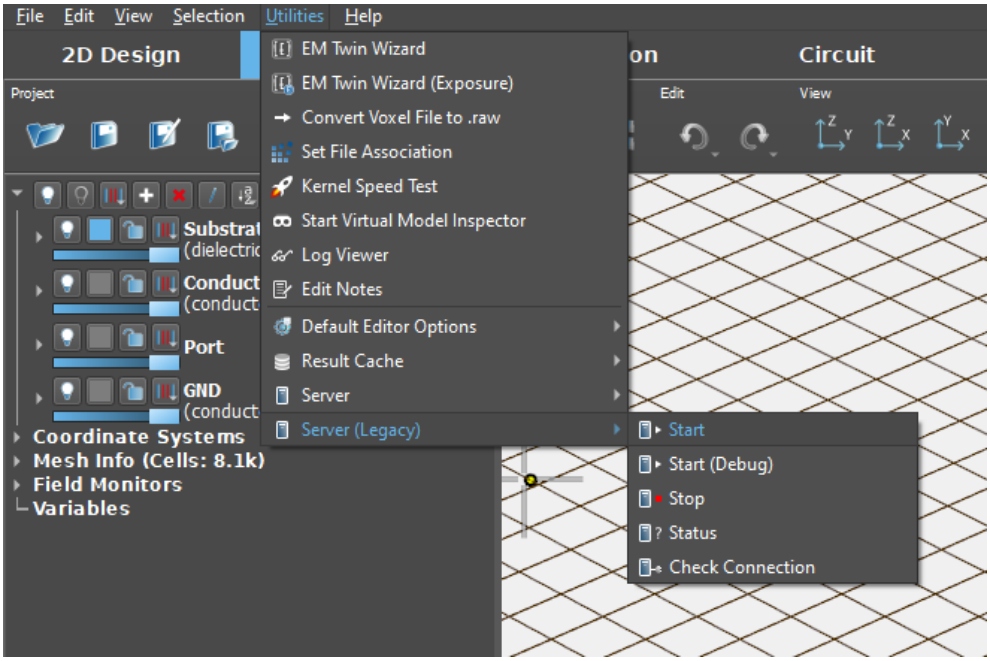


Figure 5.2.: Starting the EMPIRE XPU™ remote service from GUI

Now this machine can be found with its hostname in the **Host Setup** and can be used remotely.

Hint: Network communication

- To assure a proper communication in the network the hostnames of the server and clients have to be known. This can be tested by executing **ping hostname** on a command shell from client to server and vice versa.
- The server availability is communicated by broadcast which may not be suppressed by package filters or firewalls. If not found automatically the remote server can be added manually using the **Add** button in the **Host Setup**
- The hostnames have to be resolvable, e.g the command `nslookup hostname` should deliver the IP address and vice versa.
- Communication protocols are stored in the folder `C:\Documents and Settings\user\Local Settings\Temp` as `tmpxxxxxx.emp-log`. Please send the latest file to the support team in case of problems.

5.2.2. Remote service Windows commands

The remote service can also be started without opening the GUI. In order to start an EMPIRE XPU™ remote service as a Windows command the following scripts can be found in `C:\EMPIRE_XPU_9.x.x\empire\binW`

- EMPIRE-SERVER.cmd
- EMPIRE-SERVER-LEGACY.cmd
- EMPIRE-KILL-SERVER.cmd
- EMPIRE-KILL-LEGACY-SERVER.cmd

5.2.3. Remote service Linux commands

The remote service can also be started without opening the GUI. In order to start an EMPIRE XPU™ remote service as a Linux command the following scripts can be found in `.\EMPIRE_XPU_9.x.x\`

- EMPIRE-SERVER
- EMPIRE-SERVER-LEGACY
- EMPIRE-SERVER-STATUS
- EMPIRE-SERVER-LEGACY-STATUS
- EMPIRE-KILL-SERVER

5.2.4. Starting the remote service as a Windows service

In order to start the **legacy** EMPIRE XPU™ remote service as a Windows service the following steps have to be taken (see following subsection in case of a Windows Home edition). This is useful if the remote computing should be available each time the PC is rebooted.

- Install EMPIRE XPU™ on a Windows machine which should act as the server
- Install a license on this machine

- Either
  - Right click on **My computer** and select **Manage** or
  - Press **Start** on the Windows task bar, select **Control Panel** and open **Manage**
- Left click on **Services and Applications**
- Double left click on **Services**, see Figure 5.3
- Double left click on **Empire Service**, see Figure 5.4
- Select **Log On** and select **This account**
- Enter your username and password if applicable, see Figure 5.5 and confirm
- Confirm and return to **General**
- Set **Start up type** to **Automatic** if the service should be started each time at power up
- Press **Start** if the service should start immediately and Exit with **OK**

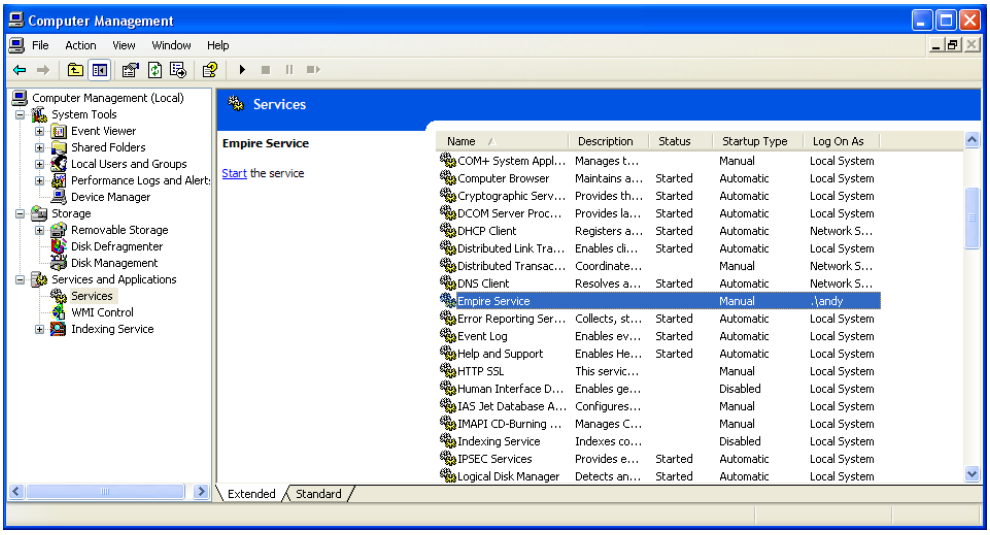


Figure 5.3.: Starting the remote server as a Windows service

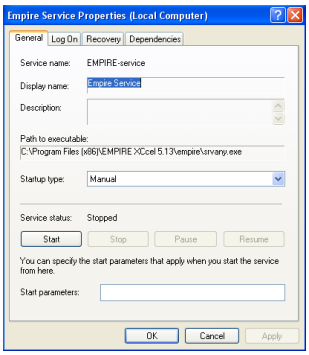


Figure 5.4.: EMPIRE XPU™ Service Properties

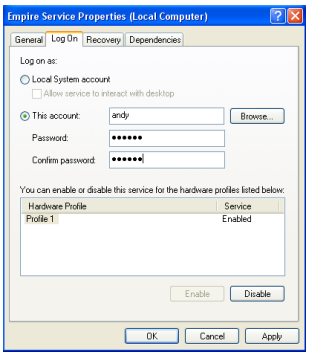


Figure 5.5.: Creating an account

5.2.5. Batch processing using IP addresses

In case if hostnames are not known in the network batch processes can also be executed using IP addresses with the steps written below.

- On server:
  - Start Empire
  - Menu: Utilities - Server (Legacy) - Start
  - Exit Empire (Verify a running python.exe with the Task Manager)
- On client:
  - Start Empire (using one example, e.g. basestation antenna)
  - Go to Simulation Tab
  - Click **Host Setup**
  - Checkmark **Enable Remote Simulation on Hosts**
  - Enter IP address for **Hosts** server and press **Add**
  - Check if server is listed and available
  - Checkmark host
  - Exit with **OK** and Start Complete Simulation

6. Multi-PC Solver

In the case if a single job requires more memory than available on a single machine the multi-PC option can be applied. This option is available to solve a single job parallel on multiple PCs within a fast network. The cluster configuration should be:

- Control PC (optional)
- Master PC (connected to the control PC with at least 100 MBit/s connection)
- Slave PCs (connected to the Master PC with at least 1 GBit/s connection[1])

Objectives

- Share one simulation job on multiple PCs within a fast network
- Reduce simulation time by usage of multiple PCs
- Simulate large structures on several standard PCs sharing the memory of multiple PCs

Usage

- The structure is set up on the control PC
- In Simulation Tab, click **Simulation Setup - Multi PC/On Disc Setup**
- **Simulation Mode: Multi PC**
- Enter number of slaves
- Click **Get hosts** and enable the *slave* PCs
- Close window with OK
- Click **Host Setup** and select *master* PC
- Close window with OK
- Select job(s) to be simulated
- Click **Start Simulation**

The control PC transfers the files to the master PC which is doing the preprocessing and compilation. After that the simulation starts and the master PC sends simulation sub-tasks to the slaves. There will be a continuous data exchange between master and the slaves using a special algorithm for the parallel execution of the simulation (the connection speed will influence the simulation performance). After the simulation the master collects the data and returns the results to the control PC.

General requirements

- All machines should run the same operating system (mixed Linux / Windows configurations are not supported)
- CPU architecture should be the same for all machines

[1] 10 Gbit/s recommended

- EMPIRE XPU™ **Legacy Server** to be started on the master and slave machines as described in Chapter 5.2
- Control PC minimum license configuration: EMPIRE-SP (Silver package)
- Master PC minimum license configuration: EMPIRE-MPC (Multi PC solver license)
- Slave PC (no license needed)
- Network Ports 45000 to 47000 are used for TCP and UDP and may not be blocked by firewall

Hint: Mutli-PC memory consideration

Because the compilation is done by the master it is recommended that this machine is configured with RAM or (SSD) disk space which can cover the complete memory required for the job. During simulation the memory needed for the main field is shared evenly between all machines. The memory needed for field dumps has to be allocated on all machines completely.

The local simulation folder for Slaves or hard disk simulation is determined by the environment variable **TEMP**.

Hint: Multi-PC Performance consideration

In contrast to Single-PC solver the Multi-PC solver algorithm needs more administration effort for the continuous exchange of data during runtime. Further, mutli-core speed optimizations can not be applied to this algorithm. Typically only half the performance of the Single-PC algorithm can be achieved and so a minimum of 3 PCs are needed for a speed-up. In addition, the performance depends on the network speed, simulation size and number of slaves.

Hint: Application Note

More information is available in Chapter 29.2.15

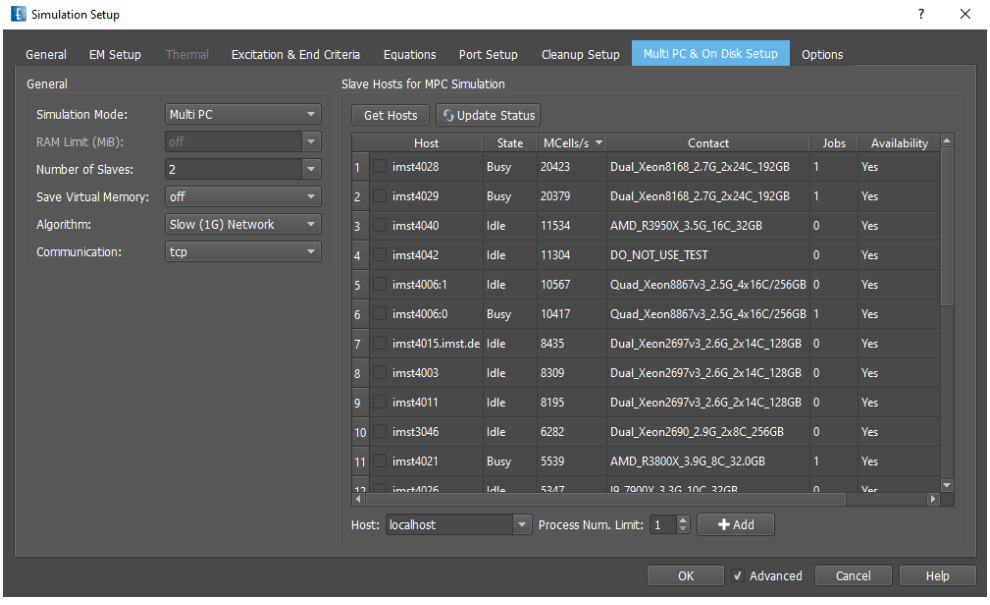


Figure 6.1.: EMPIRE XPU™ Multi-PC set up

7. Virtual Model Inspection Module

The Virtual Model Inspection (VMI) module is a new application for the HTC VIVE virtual reality system. It allows the user to move freely through scaled version of EM structures; providing a fascinating experience of its geometries from all perspectives. This enables an unprecedented layout check and design verification of simulation models, which is a crucial step before going into production. Beside the geometry, the VMI also visualizes the 3D simulation results, e.g. electromagnetic fields in different planes. Every simulation model from EMPIRE XPU™ can be virtually experienced with the VMI software.

Features:

- Model import enables the import of EMPIRE XPU™-generated 3D models in the popular .dae format
- Model inspection of the loaded models in several different sizes in a fully immersive VR environment
- 3D Field value heatmaps in three orthogonal planes provide an impressive view of the field distribution and the spatial relations to the model
- 3D Field wave propagation simulation provides a view on the propagation effects of the simulation result files
- 3D Field value mesh plot in addition to the heatmap plotting represent a standard plotting method
- In-program configuration of user options, inspector options, and graphic configuration to prevent the user from permanently switching between the VR headset and the monitor view
- In-program logging of executed commands, warnings, and errors in text form and via audio feedback
- In-program tutorial to get familiar with the basic concepts of VR experiences and interaction for users not yet familiar with VR software

Installation:

- Install the HTC™ Vive software on your PC
- Cross-check with the showroom software that SteamVR™ is correctly working
- Install EMPIRE XPU™
- Request the needed licenses
- Start the EMPIRE VMI™ in the EMPIRE XPU™ menu File-¿Start Virtual Model Inspector

Please see the complete documentation `VMI_Manual.pdf` available in the installation folder, e.g. `C:\EMPIRE_XPU_9.x.x\empire\vmi_W64`

8. Linux systems

8.1. Requirements

EMPIRE XPU™ requirements on Linux/PC Computers.

- Linux-PC/i686: e.g. Intel Core-i7/9, Xeon or AMD Ryzen , at least 4 GB Memory recommended.
- Operating Systems (64 bit)
  - Ubuntu 16.04 or higher LTS
  - RHEL/CentOS Linux 7 or 8
- 8 GB free Disk space for the EMPIRE XPU™ installation
- Graphic card with hardware accelerated OpenGL support
  - e.g. Nvidia GTX 1060
  - Latest driver version (usually available from card vendor as download, e.g, www.nvidia.com)

8.2. Installation

1. Open a Linux shell and change to the CD folder `linux`
2. Run the command: `sh ./setup.sh` and follow the instructions.
   - We do not recommend installation as super user (root) for security reasons.
   - EMPIRE XPU™ will be available to all users on the workstation from one installation by default.
   - EMPIRE XPU™ can be used via network/NFS from other workstations, too.
3. Add the EMPIRE XPU™ installation folder to your search PATH for convenience.

Remarks

- Optionally enter **1** or **2** if installation pathes should be changed
- The installation starts after entering **3** confirmed with **y**
- At the end of the installation it will be checked for missing libraries. Please install the missing libraries if indicated in the command window output.
- Optionally run the **Kernel Speed Test** to check the performance. This may take several minutes
- Enter **4** to exit the installation

8.3. Silent installation on Linux

A silent installation can be done with following procedure:

1. Open a Linux shell and change to the CD folder `linux`
2. Run the command: `sh ./setup.sh EMPIRE-INSTALL-DIR --ackn`

8.4. Start Commands

The following commands are available in the installation path, e.g. `~/EMPIRE_XPU_9.x.x`. It is advisable to add this folder to the **PATH** variable.

For license setup:

- **EMPIRE-LICENSE-REQUEST** `name.txt`
  Detects the internal id numbers and writes it into a form which can be completed and sent to `empire.support.de` to obtain a license file
- **EMPIRE-LICENSE-INSTALL** `name.txt`
  Installs the license file
- **EMPIRE-LICENSE-STATUS**
  Reports the license features which can be checked out

To start the graphical user interface and job control environment:

- **EMPIRE-XPU** or **EMPIRE**
  Starts the GUI and start-up window with access to:
  - Templates
  - Examples
  - Start from scratch
  - Open project

To test the performance:

- **EMPIRE-KERNEL-TEST** runs a small simulation and reports the performance in `MCells/s`

To run and control the remote service:

- **EMPIRE-SERVER** Reconnectable remote service
- **EMPIRE-SERVER-LEGACY** legacy remote service
  - The Empire Service will be started and can now be found or added to the host list of clients. It can used in batch and optimization jobs, provided that the machine is known in the network with its hostname, e.g. **ping hostname** or IP address.
- **EMPIRE-SERVER-STATUS** reports the status (owner, performance and user) of the reconnectable service
- **EMPIRE-SERVER-LEGACY-STATUS** reports the status (owner, performance and user) of the legacy service
- **EMPIRE-KILL-SERVER** stops the service (both reconnectable and legacy remote service)

8.4.1. Background processing

It is also possible to run EMPIRE XPU™ in the background. The commands are similar as in Windows. Assuming that `<in>` is the EMPIRE XPU™ installation folder, the command to start a simulation is:

```
<in>/empire/PY3k-64/bin/python -O -u <in>/empire/env.py simul d.emx
```

8.5. License Request

On Linux systems the license can only be issued to network card MAC id numbers of the Ethernet Interface (12 hexadecimal digits). The licensing software reads the number of the network card which supports a certain naming convention.

Hint: Linux Ethernet Interface Name

It is necessary that the network card supports the following naming policy:

- For node-locked licenses the udev/systemd is supported, (e.g. eno1, ens1, enp2s0, eth0) with a maximum length of 24 characters
- For floating licenses the network card name of the license server has to be **eth0**.

The name of the network card can be checked with the command `ifconfig -a`. It is possible to change the name of a system.

For creating the license request file:

- Please follow the license request guide in section 2.4.
- Alternatively, run **lmhostid** in `/EMPIRE_XPU_9.x.x/empire/flexlm/linux`

8.6. Install the License

- Please follow the license install guide in section 2.5.
- Alternatively, copy the license file into `/EMPIRE_XPU_9.x.x/empire/flexlm` and rename to **license.dat**

8.7. Floating license

This section describes the steps for using a Linux machine as a license server.

8.7.1. Install Programs

It is not necessary to install the complete EMPIRE XPU™ on this server, the following programs which can be found in `\empire\flexlm\winnt\linux` are sufficient:

- **START_FLOATING_LICENSE** Script to start the service
- **STOP_FLOATING_LICENSE** Script to stop the service
- **lmstat** license diagnostic
- **lmgrd** License Manager Daemon

- **imstlmd** Vendor Daemon
- **lmdhostid** Return id number
- **README.txt** Prerequisites for license service on Linux

If the command **./lmhostid** does not work the following packages have to be installed:

- Ubuntu (12.04 or higher): sudo apt-get install lsb-core
- Fedora (core 15 or higher): sudo yum install redhat-lsb

8.7.2. Check Interface Name

If the command **./lmhostid** yields only zero digits the interface name has to be renamed to **eth0**

- Get current name of ethernet interface with **ifconfig -a**, (e.g. enp0s25)
- Rename to **eth0**
  - Create a link file `/etc/systemd/network/01-eth0.link`
  - Edit content:

`[Match]`	
`OriginalName=enp0s25`	Use current name
`[Link]`	
`Name=eth0`	Set to new name

  - Run `sudo udevadm trigger --action=add`

Now the command **./lmhostid** should yield ether address as displayed by **ifconfig -a**

8.7.3. Start License Server

To start a Linux system as a license server the license manager daemon **lmgrd** has to be started. This program invokes the **imstlmd** vendor daemon. The status of the license server can be displayed with the **lmstat** executable.

The current version of lmgrd is 11.11.1 (Linux Standard Base(TM) certified, May 2012) and supports all recent 64 bit Linux system. Older versions of lmgrd are supplied, too, for compatibility reasons. Newer versions may be found on the web. All versions of lmgrd should be compatible with the vendor daemon **imstlmd**.

A small script called **START_FLOATING_LICENSE.sh** is available, too, which can be used to start the license server instead of starting lmgrd.

The script does the following:

- Creates a folder **empire_license_logdir** in the home directory, which name can be edited
- Creates backup files from old log files with a time stamp
- Starts the **lmgrd** daemon
- Redirects the log files into the above mentioned folder

8.7.4. Delayed license checkout (Linux)

Similar as in section 4.2.1 a broken TCPIP connection can lead to a delayed license checkout for used features. On Linux systems the **Keep Alive Time** can be set similar as follows:

Assuming the shell ksh is used the command `ndd -set /dev/tcp tcp_keepalive_interval $((120 * 1000))` will set the keep alive time to 120 seconds, only.

See "man ndd" for more info.

8.8. Trouble Shooting

At the end of the installation possible missing libraries or conflicts are detected. If required additional packages have to be installed manually to run EMPIRE XPU™. In case of troubles, the command **EMPIRE-CHECK-INSTALL** can help to detect and resolve further problems.

Graphics

EMPIRE XPU™ needs an OpenGL graphic card and a fitting driver for this. The latest driver should be downloaded and installed which can be obtained from the card vendor. If there are still problems the output of the command **glxinfo** should be sent to the EMPIRE XPU™ Support.

If no OpenGL is available EMPIRE XPU™ can be used restrictively by executing the command **EMPIRE-XPU -nogl**

C-Compiler

The installation of a `C/C++`-Compiler is a prerequisite. This is usually not available in a standard configuration and has to be installed from the Linux Distribution.

Dynamic libraries

EMPIRE XPU™ has been tested on Ubuntu and Redhat Linux and are the recommended platforms. If other Linux distributions have to be used, some shared libraries can be missing. Usually, the missing libraries can be found in a Package Manager.

If an error message appears with an invalid command name, one can search for this file and check it's dependencies using the **ldd** command. After installing the reported missing libraries the problem can be fixed.

As an example the following packages need to be installed on a fresh CentOS 8 system (using `sudo yum install`):

- **glibc-headers**
- **mesa_libGLU**
- **kernel-devel**
- **glibc-devel**

Incidents

In case of the following incident

```
ImportError: /lib64/libhogweed.so.4
```

the following command in the installation folder helps

```
mv empire/PY3k-64/lib/libgmp.so.10 empire/PY3k-64/lib/libgmp.so.10.b1
```

9. Data Structure

9.1. Input Files

With **Save** or **Save As** the model (structure, simulation parameters and setup) is stored as **name.emx**. It automatically creates a project folder **name** which contains support files, e.g. **name.par** for simulation parameters and a backup file **name.gbk** as a copy of the **.emx** file of the next to last save action.

If a pre-processing is executed, the structure is discretized and prepared for simulation in a file called **name.acad**. Pre-processing statistics are written in a file **name.log**. The FDTD kernel uses the **.par** and **.acad** files as input for the simulation.

Hint: Network drives

In order to simulate projects on a network the project folder has to be located on a mapped network drive containing a drive letter, e.g. `X:\`

Hint: Folder names

Special characters (non-UTF8 letters) should be avoided in the folder and project names.

9.1.1. File Recovery

In order to recover the input file from the last save state rename the backup file **name.gbk** to **name.emx**

9.1.2. Compatibility

Input files (ending **.gym**) which were created with version 7.72 and older can be loaded with the current version. They are labelled as **Legacy Project** in the header line until it is saved into a new folder.

Input files (ending **.emx)** which were created with the current version can only be loaded with version 7.72 by renaming the ending to **.gym** and placing the file into the project folder.

9.2. Simulation Files

If the simulation mode is not **Simultaneous**, result folders **sub-k** are created during the simulation for each exciting port number **k** and results are stored inside.

In case of parameter sweeps or optimizations **v-opt-...** folders are created for each parameter combination and the result folders **sub-k** are placed inside.

During simulation, time domain result files are created and updated frequently (e.g. voltages and currents of each port). At the end of the simulation (or during the simulation in the event of an output synchronization) field monitor dumps are created.

Hint: Opened files

The number of opened files is limited and depend on the operating system (Windows: 200, Linux: 800). If this number is exceeded files have to be closed and reopened for each update which can drop down the simulation performance in case of many ports, especially on a network drive.

9.3. Postprocessing Files

After simulation an automated post processing is executed in each result folder which creates Frequency or Angular (far field) domain result files. With the exception of field dumps the result files are in ASCII format which are explained in Chapter 25

10. High Performance Computing

10.1. Single machine consideration

EMPIRE XPU™ is highly optimized for fast simulation and utilizes many hardware features:

- **SSD** A solid state disk is recommended for a quick start of the GUI and local simulation. Further, EMPIRE XPU™ features **On-disc** simulation to extend the physical memory which is fastest using **SSD**.
- **Cores** By default all available cores are used and the simulation runs parallel performing muliple time steps inside their caches. The performance can drop if
  - other programs (e.g. Remote Desktop graphics) run at the same time. If required, the number of cores (threads) can be limited in the **Simulation Setup - Options**
  - not enough cells are available in z-direction ($N_z > 3N_{cores}$ should be fulfilled). It may help to rotate the strcuture to meet this condition.
- **CPUs** Parallel simulation is also supported on multiple CPUs. Best performance is obtained if they are connected with QPI (Quick Path Interface)
- **AVX** Advanced Vector Extensions allow the parallel execution of floating operations and is supported by default (also MMX, SSE, AVX-2, ...). Its usage is printed in the fdtd.log file.
- **NUMA** Non uniform memory access can accelerate simulations and should be enabled in the BIOS setting. If required (e.g. running multiple EMPIRE-SERVERS on one machine) the number of NUMA nodes can be limited on Linux systems by defining an environment variable, e.g. `export EMPIRE_NUMA_NODES=0-1` for using only 2 NUMA nodes.
- **Hyperthreading** is only a virtual enlargement and can not be exploited if cores are busy during FDTD computation. It can be switched off in BIOS without loss of performance. Note, if switched on only half of the load is displayed in the Windows task manager even if all physical cores are busy.
- **On Disk Simulation** If the RAM is not sufficient this option can be enabled in the Simulation Setup to increase the memory. Best performance is obtained by using a Solid State Disk.
- **Multi-PC Simulation** If the structure is large enough the performance can also be increased and the memory can be shared by using a master and several slave PCs (with similar architecture) in parallel.

In general the simulation performance is determined by

- CPU clock speed
- Number of cores
- Last Level Cache size
- CPU vector extensions (e.g. MMX, SSE, AVX, ...)
- Memory Bandwidth (all memory sockets should be equipped)

The peak performance can be obtianed by running the **Kernel Speedtest** available in the **Utilities** menu. The kernel speedtest runs an empty model (vacuum) with electric boundary conditions and no field monitors. All available cores are used and usually this value can be used to compare the performance on different machines.

10.2. Multiple machine considerations

For invoking multiple machines there are basically 2 possibilities:

10.2.1. Multiple jobs on different machines (job distribution)

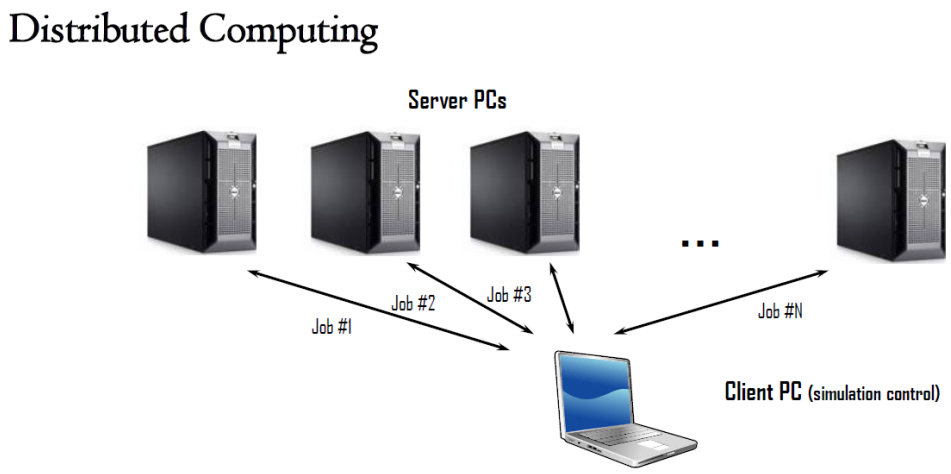


Figure 10.1.: Job distribution (using several Single-PC Solver licenses)

- One machine acts as control PC which
  - distributes the jobs to the different machines and uploads the input files to the remote hosts
  - receives and displays status information in a log window
  - collects the result files from the remote hosts
  - executes the post processing
- Machines may be heterogeneous
- Star-like data exchange
- A ”Single-PC solver” license is needed for each machine
- Memory:
  - The compilations are done sequentially and the memory will be reused
  - During simulation the memory for each job is limited by individual machine

10.2.2. Single jobs on different machines (job sharing)

- One machine acts as the master PC which does the compilation
- Several other machines act as slaves which share the simulation job with the master
- Machines should be equal in CPU architecture and operating system
- Ring-like data exchange

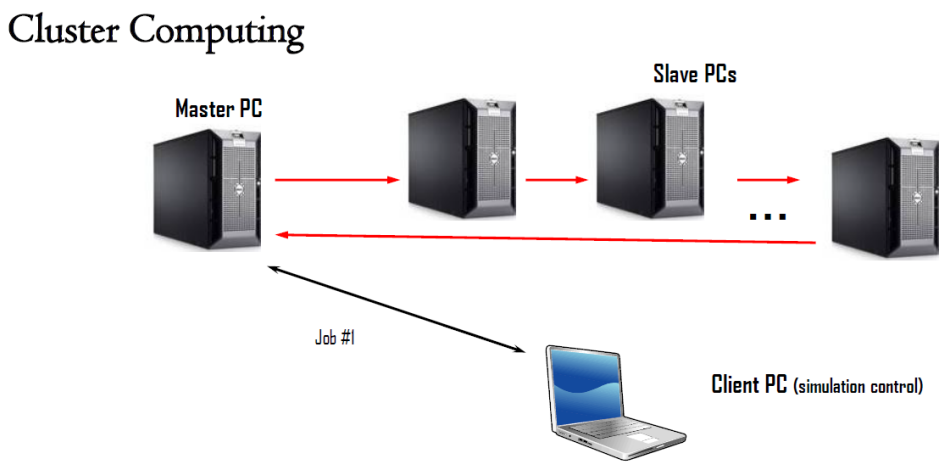


Figure 10.2.: Job sharing (using one Multi-PC Solver license)

- Performance depends on model size, connection speed, single machine performances and number of machines (there will be a saturation at some point)
- A ”Multi-PC FDTD solver” license is needed for the master
- For the optional client PC a ”Frontend” license is needed
- Memory:
  - During compilation the memory can not be shared. The master should either be equipped with large memory (the hard disk support is enabled if more memory is needed)
  - During simulation the memory for the FDTD algorithm can be shared but the memory for field recordings have to be allocated on each machine

Note the performance considerations in Chapter 6. Especially for machines with large number of cores a speed-up is only possible if the model is very large and the network speed is very high at the same time.

11. Update & Support

11.1. Version Control

The software version can be identified by the unique **Build** number which is

- available as a file in the software distribution, e.g. **B23248.txt**
- mentioned through the installation procedure
- displayed in the top left area and **Help** - **Info** Menu
- printed in simulation (fdtd.log) and input files (.emx)

Further a Release number is used for licensing and version and revision control, e.g. 9.1.0: Version 9.1, Revision 0.

- The Version name is used in the installation path by default. It is possible to install different versions on a PC.
- Licenses are created for the Version number and are valid for different revisions
- Licenses are down-compatible to lower version numbers

Updates will be announced on the EMPIRE XPU™ webpage along with the change documentation.

11.2. Web Sites

- Updated information about EMPIRE XPU™ can be found on **http://www.empire.de/**
- Information about our company can be found at **http://www.imst.com/**

11.3. Contact Us

- We highly appreciate any comment to EMPIRE XPU™. If you have any comment please let us know by sending an email to **empire.support@imst.de.**
- If you have any trouble, do not hesitate to contact your local support team or our German support team **empire.support@imst.de**
- If we do not reply within two working days, assume a transmission fault of the data. Please retry to contact us.
- Please notice our general working hours Monday to Friday from 9:00 to 17:00 CET, except for German public holidays.

11.4. Contact Local Support Teams

In several countries, local EMPIRE XPU™ support teams can give a fast response in your time zone.

- North America : `empire.support.us@imst.com`
- Europe : `empire.support.eu@imst.com`
- Japan: `empire.support.jp@imst.con`
- Germany and other countries: `empire.support@imst.com`

Part II.

Empire User Interface

12. Introduction

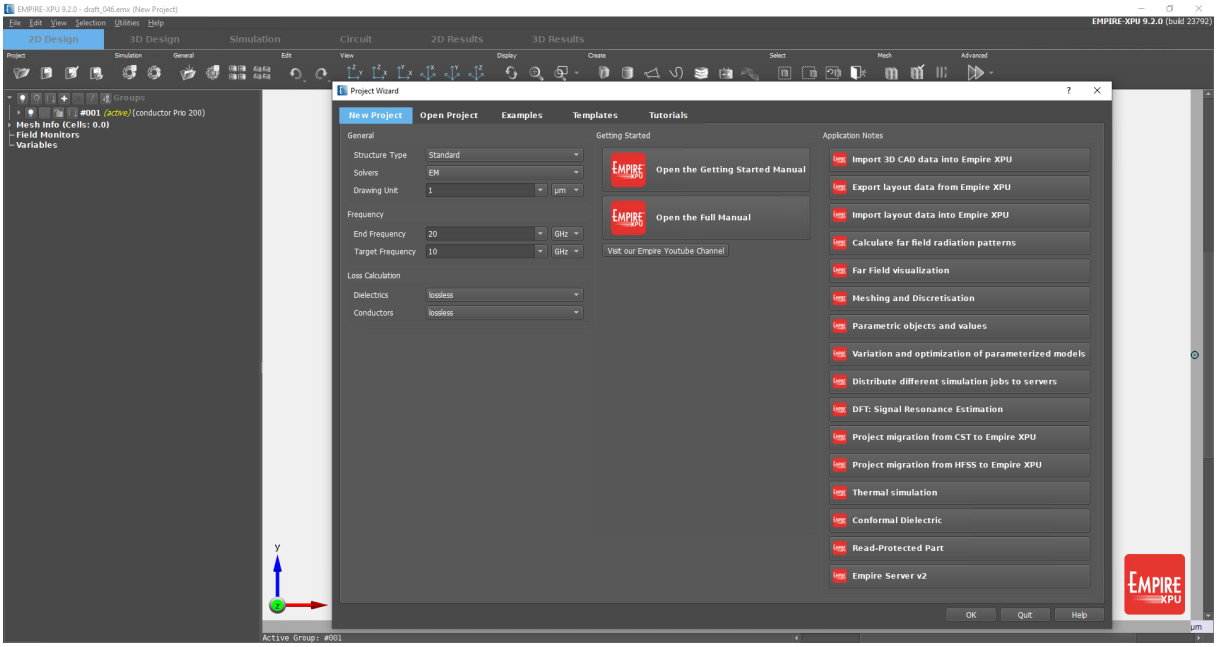


Figure 12.1.: EMPIRE XPU™ start-up window

This chapter gives an introduction to the EMPIRE XPU™ user interface and refers to more detailed chapters which deal with basic and advanced features. In this chapter the different window areas will be explained.

By starting the EMPIRE XPU™ icon from the desktop, the window as in Figure 12.1 appears if a valid license file has been installed as explained in installation part. Here, global parameters which are specific for the current project can be entered. These parameters will be explained in chapter 23.

- If **New Project** is selected EMPIRE XPU™ is started from scratch and the window as displayed in figure 12.2 is opened. By clicking on a button in sections **Getting Started** or **Application Notes**, hyperlinks connect to manual documents or the EMPIRE XPU™ You Tube channel with a collection of video tutorials.
- It is also possible to open an existing project by clicking the **Open Project** Tab and select one of the recently used files.
- In **Examples** a collection of already simulated projects with results is available which gives a quick overview of different applications and features.
- With **Templates** many typical configurations can be selected which design parameters are adjustable before opening. A comprehensive description of the templates is available in chapter 27.
- In **Tutorials** step-by-step instructions are available as PDF files for many applications to learn the simulation flow of EMPIRE XPU™ from scratch to results

In the following sections the different window areas are explained in detail.

Hint: Right Click on item

With a right mouse click on many items (e.g.list entries, selected objects, monitors) a context menu is displayed which contains a set of possible actions.

Hint: Tooltips on items

By hovering the pointer over an item (e.g. button, input field, switch) a description is depicted with further explanation and help

Hint: Drag and Drop

With Drag and Drop (left click and hold list entry and drag to other list entry) some items (e.g. groups) can be moved

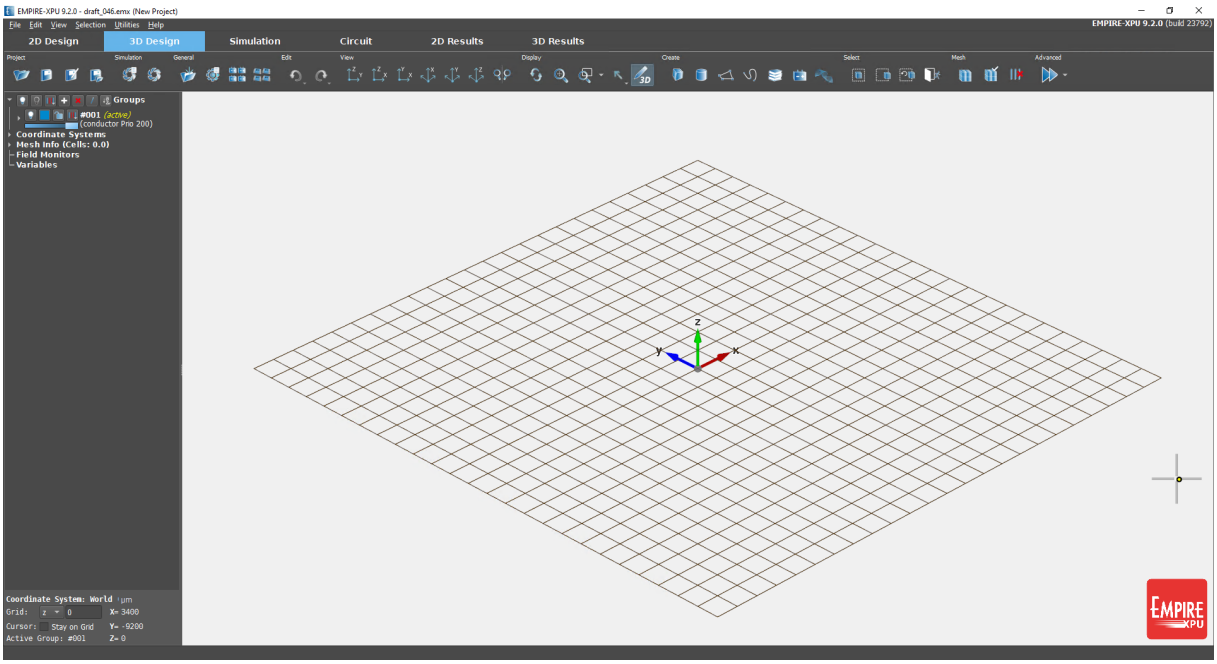


Figure 12.2.: EMPIRE XPU™ started from scratch

12.1. Top tool bars

The top menu and top left icons in the EMPIRE XPU™ window define the standard commands which are always available, like save, import, export, options, views etc.

12.1.1. Menu items

Menu items (Figure 12.3) are located in the first row.

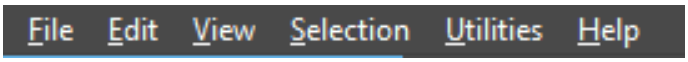


Figure 12.3.: EMPIRE XPU™ menu tool bar

File Menu

- **New Project** Begin a new project
- **Open Project** Open an existing project by loading a file ending .emx
- **Recently Used** Displays a list with recently used project files to be selected
- **Save** Save current project file
- **Save as** Copy current project input and optional support files and open new file
- **Save as (with Results)** Copy current project file including results and open new file
- **Store Variant** Copy current project folder into a sub folder and stay in opened input file. A copy of the current input file is saved in legacy format **.gym**
- **Import** Import structure in different formats
- **Export** Export structure or image in different formats
- **Run Script** Advanced: Execute python script
- **Reload external Script** Advanced: Updating data base if script has been modified
- **License** Request and installation of licenses
- **Exit Empire** User will be prompted for saving and closing the simulation engine

Edit Menu

- **Undo** Undo last action
- **Redo** Redo action which has been undone
- **Copy** - Ctrl c - Copy selections to clipboard
- **Cut** - Ctrl x - Cut out selections and copy to clipboard
- **Paste** - Ctrl v - Paste selections into current project after entering a shift vector
- **Repeat Last Action** Apply latest action
- **Search Action** Browse through a list of all available actions

View Menu

- **Redraw** Update screen
- **Zoom Extents** Zoom out using bounding box of all visible objects
- **Side, Front, Top, Iso, Opposite** A collection of predefined normal and isometric views (camera positions)
- **Save, Restore View** Store and recall current camera position
- **Advanced Normal, Advanced Iso** A collection of predefined views (camera positions)

Selection Menu

- **Select All** Select all visible and non-locked objects, shortcut **Ctrl-a**
- **Deselect Last Selected**, unselect last entry, shortcut **Backspace**
- **Deselect First Selected**, unselect first entry, shortcut **Shift+Backspace**
- **Internal Clipboard** Selections and operators can be stored in function keys **Shift F1, F3, F4**, and recalled with **F1, F3, F4**. Useful if operators, like points or arrows, should be used several times in subsequent actions.

Utilities Menu

- **EM Twin Wizard** Start wizard for a CAD import and placing an antenna digital twin field source on the CAD surface. See Chapter 3.1.1 of **Getting Started EM-Twin.pdf** for more details.
- **EM Twin Wizard (Exposure)** Start Wizared for the exposure investigations, where incident or absorbed power density or SAR can be determined using an antenna digital twin field source. See Chapter 3.1.2 of **Getting Started EM-Twin.pdf** for more details.
- **Convert Voxel File to .raw** Stores voxel models (ending .vxx) into .raw format
- **Set File Association** Associates files ending .gym or .emx to the current version
- **Kernel Speed Test** Returns the performance figure of the local machine in MCells/s
- **Start Virtual Model Inspector** Virtual Reality support of HTC Vive glasses
- **Log Viewer** Browse current project folder and open .log files
- **Edit Notes** Open the **Notes Editor** for storing userdefined notes along with time stamps
- **Default Editor Options**
  - **Save** Save settings which are modified in the Editor Options
  - **Load** Load saved settings
  - **Restore** Recall the default settings
- **Result Cache** Accelerate result update in **2D Results** plots by limiting file search.
  - **Create & Enable** Use result cache. File search is limited.
  - **Delete & Disable** Clear cache. Complete file search is activated on each data reload
- **Server** EMPIRE XPU™ (reconnectable) remote service
  - **Start** the server on the current machine
  - **Status** of the server on the current machine
  - **Stop** the server on the current machine
- **Server (Legacy)** EMPIRE XPU™ remote service (Client - Server connection needed during simulation)
  - **Start** of the server on the current machine
  - **Start (Debug)** Sart the server on the current machine and create a log file on the server machine for debug purposes
  - **Stop** of the server on the current machine
  - **Status** of the server on the current machine
  - **Check Connection** with a server on a remote machine. A log file is created in the current project folder.

Help Menu

- **Info** Display software information see Figure 12.4
  - **About Empire** Short description and link to EMPIRE XPU™ homepage
  - **About IMST** Short description and link to IMST homepage
  - **License Agreement** Show software license policy
  - **License Info** Check status of available license features
  - **System Info** Show Version and Build number and more information about the active graphic card if **3D Design** tab is selected
- **Manual** Open manuals and application notes in PDF format in a browser
- **Scripting (KMF)** Information on the scripting interface for advanced users
- **GUI Configuration** Select language package and set size of e.g. icons and menu fonts
- **GUI Reset** Reset GUI layout to default values
- **About Empire-XPU (Web)** Link to EMPIRE XPU™ homepage
- **About IMST (Web)** Link to IMST homepage



Figure 12.4.: EMPIRE XPU™ Help - Info window



Figure 12.5.: EMPIRE XPU™ tabulator tool bar

12.1.2. Application tabs

Below the menu items there **Tabs** (Figure 12.5) located to switch between different applications:

- **2D Design** This edit mode is optimized for planar structures and displays objects on top of each other in a wire frame mode. Object creation and modification is entered in a 2D view, either xy- (top, default), xz- (front) or yz- (side) planes. This mode uses the group height for object creation by default. The current mesh is displayed on side bars.
- **3D Design** This edit mode is intended for general use and displays objects in 3D view. Object creation and modification is entered in a 3D view. Further, local coordinate systems u,v,w can be defined and grids x=const, y=const, and z=const are displayed to access coordinates.
- **Simulation** Set up of advanced simulation parameters, remote control, and post-processing (e.g. user defined equations) as well as advanced simulation such as multi-PC, batch-processing, parameter sweep and optimization.
- **Circuit** Schematic editor for lumped circuit elements. Results from EM simulation or measurements can be embedded as an S-Matrix or Touchstone File and postprocessed in frequency domain.
- **2D Results** Display of 2D plot results, e.g. Scattering Parameters vs. frequency, Voltage vs. time, Far field vs. angle, E-field vs. length, etc.
- **3D Results** Here, the structure is displayed in a rendered view and can be scaled and sliced. Mesh, Boundaries, near and far fields can be visualized.

12.1.3. Top left icons

Below the application tabulators the following top left icons are are permanently available (Figure 12.6).



Figure 12.6.: EMPIRE XPU™ icon tool bar

- Project
  - **Open** Open another project. User will be prompted to save if changes have been made.
  - **Save** Save the current project.
  - **Save As** Save the current project in a new file and open it.
  - **Store Variant** Save current project in a new sub folder and keep the current file opened.
- Simulation
  - **Simulation Setup** Open the Simulation Setup window 12.7 for entering simulation parameters
  - **Simulation Control** Start a standard simulation with the current input file.
- General

  - **Import** Open a file select box for the import of external data, such as layouts or 3D structures.
  - **Editor Options** Open the Editor Option window 12.8 for frequently used settings
  - **Port Setup Wizard** Open the port array setup for the quick access to port properties (the port array arrangement is dependent on the current view)
  - **SMD Setup Wizard** Open the SMD array setup for the quick access to SMD components
- Edit
  - **Undo** Undo last action(s).
  - **Redo** Redo action(s) which have been undone.
- View
  - **Side view** Display the yz plane
  - **Front view** Display the xz plane
  - **Top view** Display the xy plane
  - **View Iso x** 3D view where lines parallel to x axis are displayed vertically.
  - **View Iso y** 3D view where lines parallel to y axis are displayed vertically.
  - **View Iso z** 3D view where lines parallel to z axis are displayed vertically.
  - **Opposite View** Reverts the current view (only in 3D)
- Display
  - **Redraw** Refresh window
  - **Zoom extends** Display maximum area defined by objects created so far and current grid.
  - **Zoom window** Display area defined by an arrow
  - List of more zoom options
  - **3D Mode** Choose cursor mode (only in 3D)
  - **3D Input** Toggle coordinate input mode (with or without grid) (only in 3D)
- Select
  - **Select enclosed** Select all objects within an area
  - **Select overlapping** Select all objects overlapping an area
  - **Select last modified** Select last modified object
  - **Escape** Clear selections and cancel actions

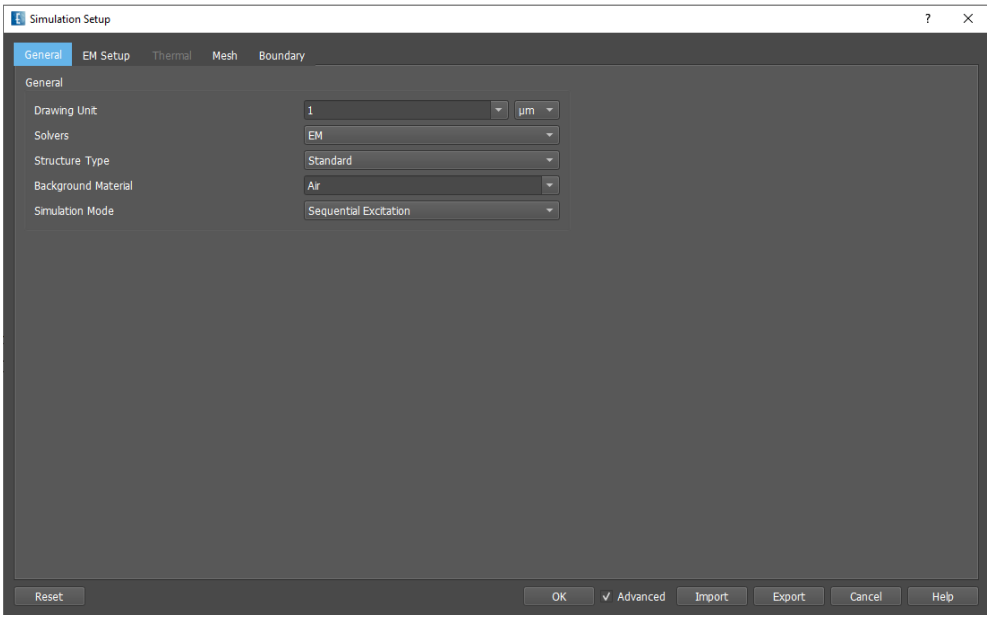


Figure 12.7.: EMPIRE XPU™ Simulation Settings

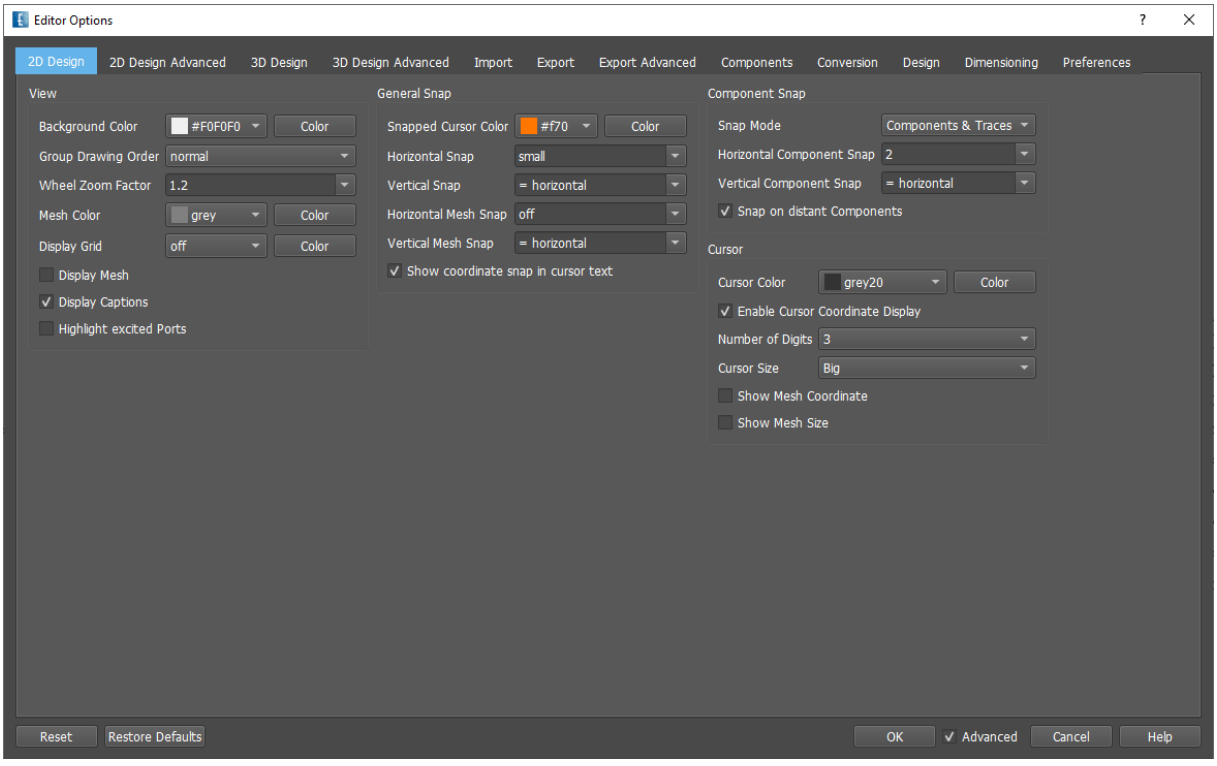


Figure 12.8.: EMPIRE XPU™ Editor Options

12.2. 3D Design Mode

The **3D Design** mode (Figure 12.9) is intended for editing the structure in 3D view. This mode is best suited for

- Relative 3D positioning of objects and ports
- General 3D structures
- Objects defined in local coordinate systems (LCS)

Typical operations are:

- Create objects and ports in 3D view (Create buttons)
- Select objects (Left click on items)
- Enter arrows for shift, copy, stretch or mirror (Drag left mouse)
- Enter points for alignment, rotate (Shift + left click)

In this mode, the cursor coordinates are derived either

- from the grid, e.g. x=const, y=const, or z=const
- from the object's visible surface

The latter can be disabled by checkmarking the **Cursor: Stay on Grid** in the bottom on the left. At any time the user can switch to the **2D Design** mode and back again.

12.2.1. Grid

By default the background is white with a grid in grey. The settings can be changed in the Editor Options - 3D Design and 3D Design Advanced.

- The grid spacing depends on the current zoom factor and displays rounded coordinates. It can be changed in Editor Options - 3D Design Advanced - Grid Resolution
- By default, the grid displays x,y coordinates. It can be switched to y,z and z,x by pressing the **Space** bar or entering, e.g. x=100 in the left bottom. In Local Coordinate System (LCS) u,v,w coordinates are used.
- More grid options can be set in the **Context Menu** (Figure 12.10) (click right mouse button in workspace)
- The cursor snaps on the grid coordinates or object edges, vertices and midpoints
- The grid can be deactivated by pressing the grid button in the right bottom 

In order to approach a certain coordinate x=x0, y=y0, z=z0, move the cursor to x0,y0, press the space key, and move the cursor to y0, z0.

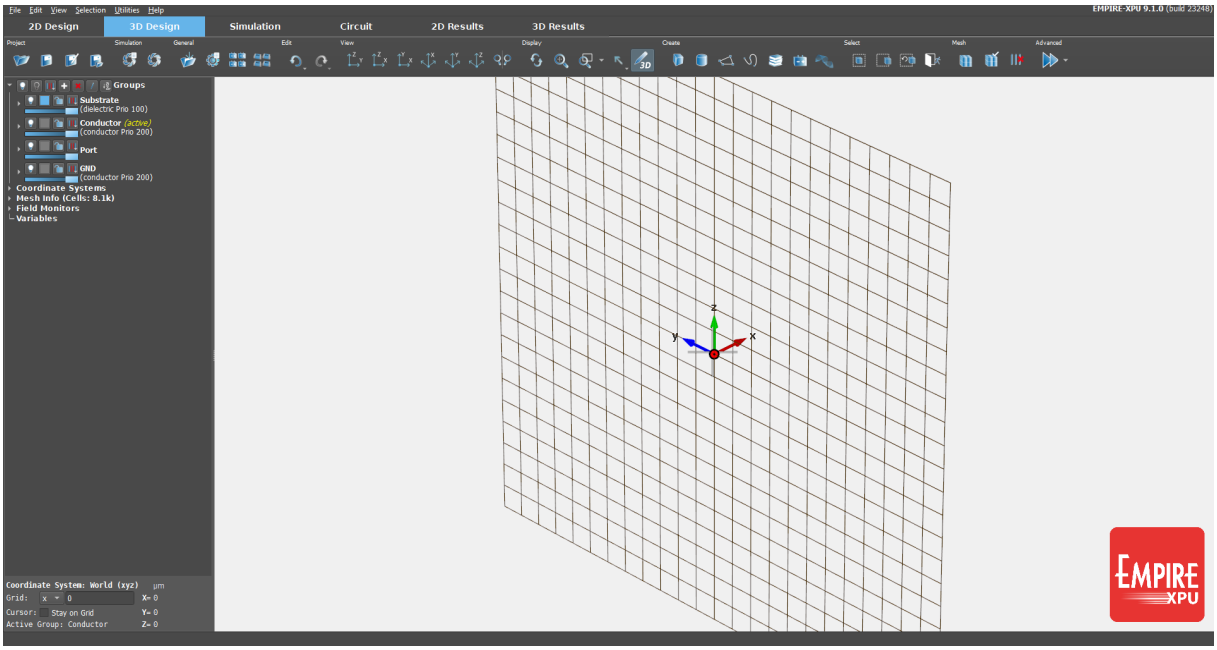


Figure 12.9.: 3D Design mode display showing tripod and yz grid lines at x=0

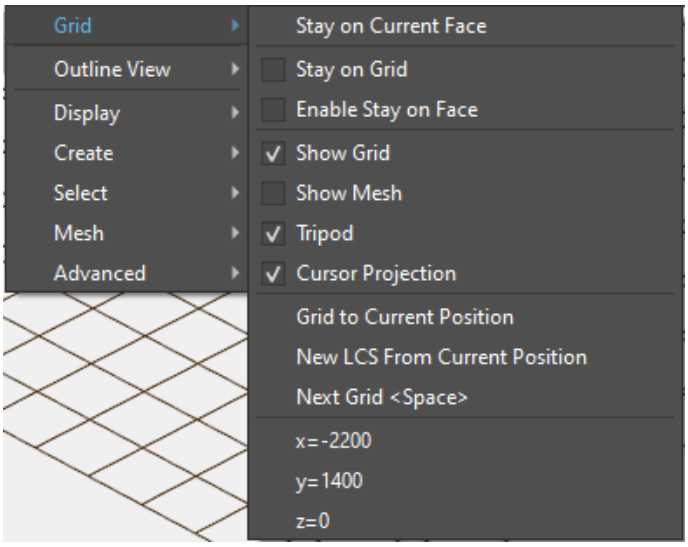


Figure 12.10.: EMPIRE XPU™ Context Menu (right click in workspace)

12.2.2. Cursor

The current cursor position is displayed by a large cross. A dot snaps to rounded coordinates or object locations. Depending on the snapped position the appearance of the dot changes:

- Cursor snapped to origin
- Cursor snapped to rounded coordinate on grid
- Cursor snapped to rounded coordinate on object face
- Projection of cursor on surface to grid
- Cursor snapped to rounded coordinate on object edge
- Cursor snapped to object vertex
- Cursor snapped to object center
- Cursor snapped to object curvature showing radius and angle

The snap options can be adjusted in the Editor Options Menu and Context Menu.

12.2.3. Coordinate Systems

In 3D Design mode coordinate systems are listed in the navigation tree on the left. With a right click on this entry coordinate systems can be created:

- **Add Coordinate Systems** LCS with editable P0 and u,v,w Matrix. This LCS can also be created with a right click on a (curved) object and selecting **Grid - Get LCS from Face**
- **Add Transform Coordinate Systems** LCS with a list of transform operations
- **Delete Unused Coordinate Systems** Remove LCS which are not referenced by any object

If several coordinate systems are available only one can be active. A coordinate system can be activated with right click and **Set Active**. The **View** buttons at the top apply to the active coordinate system.

WCS

The world coordinate system (WCS) is defined by x,y,z coordinates and displayed in the left bottom.

- Only one WCS can be defined
- The Origin of WCS can be changed by entereing a point and applying the function **Change origin of Word Coordinate System**
- The WCS is displayed with a tripod labelled as xyz

LCS (Transform)

A transformed local coordinate system (LCS) with u,v,w coordinates is defined by a sequence of transformations: shift, rotation and scale with respect to the WCS. The LCS origin and orientation is displayed by the u,v,w tripod and can be defined by a right click:

- Navigation tree - Coordiante Systems - Add Transform Coordinate System

With a right click on the LCS transformations can be added (variables may be used)

- Add Transform **Translate** Shift vector in units
- Add Transform **Rotate U** Turn vw plane with angle in degrees
- Add Transform **Rotate V** Turn wu plane with angle in degrees
- Add Transform **Rotate W** Turn uv plane with angle in degrees
- Add Transform **Scale** Relative scale vector

The sequence of the transformations is displayed as in (Figure 12.11). Additional transformations can be entered before or after existing transformations. Values may be edited by double-click on the respective entries.

LCS

A local coordinate system (LCS) with u,v,w coordinates is defined by a point and rotation matrix with respect to the WCS. The LCS origin and orientation is displayed by the u,v,w tripod and can be defined by:

- Navigation tree - Coordiante Systems - Add Coordinate System
- Move cursor to object face, right click: Context menu - Grid -**New LCS From Face Position** (Figure 12.10)
- Move cursor to grid, right click: Context menu - **New LCS From Current Position** (Figure 12.10)

A rotation matrix is displayed as in Figure 12.11 which values may be edited by double-click on the respective entries. With a right click on u,v,w rotation or mirror operations can be applied to change orientation.

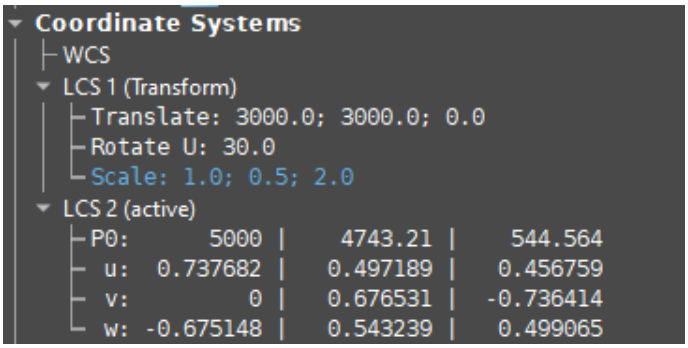


Figure 12.11.: Example of Coordniate Systems

All LCS can be named and activated by selecting the LCS in the Navigation Tree - Coordinate Systems.

Hint

If an LCS is deleted, all objects which are defined in this LCS will be deleted, too.

12.2.4. Zoom

In the **3D Design** mode the drawing area can be zoomed by

- Using the Top icons 
- Using the mouse wheel
- Using the **Page up** / **Page down** keys on the keyboard
- With Shift and wheel a lens is activated at cursor position. Use **Esc** to deactivate

Hint

Use **Zoom extends** button  or the **z** shortcut on the keyboard to obtain the total view

12.2.5. Pan

In the **3D Design** mode the drawing area can be shifted to the sides by

- Dragging the middle mouse button
- Using the arrow keys ↑←↓→ on the keyboard
- Clicking the **Choose 3D mode: Pan**  Button and dragging the left mouse button

12.2.6. Rotate

In the **3D Design** mode the structure can be rotated by

- Dragging the right mouse button
- Clicking the **Choose 3D mode: Rotate** Button  and dragging the left mouse button

By dragging the mouse button in horizontal direction, the structure is rotated around the vertical axis and vice versa. By using the wheel while holding the right mouse button, the rotation is done around the orthogonal axis.

Hint

The axis of rotation is derived from the current cursor position, e.g. point on the object surface. If the cursor is far apart from the object the rotation can lead to unexpected views. Use **Iso z view** button  and **Zoom extents** to obtain the standard view.

12.2.7. Bottom line

On the left bottom there are fields for

- Used unit, e.g. **um** for microns
- Current coordinate system
- Checkmark selection for keeping cursor on grid instead on object surface
- Grid plane
- Cursor coordinate in World Coordinate System (WCS) (x,y,z)
- Cursor coordinate in Local Coordinate System (LCS) (u,v,w)
- Active group
- Number of selections and selected items

12.2.8. 3D Outline View

By default, object's faces are displayed in a color filled mode with black edges. With a right click the **Outline View** can be changed:

- Edges only (wire frame)
- Color filled faces, edges black (default)
- Color filled faces, edges colored
- Color filled faces, no edges

12.3. 2D Design Mode

The **2D Design** mode is intended for editing the structure in 2D view. This mode is best suited for

- Exact 2D positioning of objects and ports
- Planar and multi-layered structures
- Inspection of structure with respect to mesh and boundaries
- Display and positioning of field monitors

Typical operations are:

- Create objects and ports in 2D plane (Create buttons)
- Select objects (Left click on items)
- Enter arrows for shift, copy, stretch or mirror (Drag left mouse)
- Enter points for alignment, rotate (Shift + left click)

By default, this mode displays the xy-plane which is the Top View  of the structure to be entered. To switch to the xz-plane (Front view)  or the yz-plane (Side view) viewport icons are available in the top tool bar.

Hint: Layout import

Before importing 2D layout data switch to the desired drawing plane.

12.3.1. Background

By default the background is white and blank. It can be changed in the Editor Options - 2D Design:

- View - Background Color
- View - Display Mesh in drawing area (see 12.12)
- View - Display Grid (equidistant) in drawing area

12.3.2. Zoom

In the **Draft** mode the drawing area can be zoomed by

- Using the Top icons 
- Dragging an arrow (left mouse button) and using the **z** shortcut on the keyboard
- Using the mouse wheel
- Using the **Page up** / **Page down** keys on the keyboard

12.3.3. Pan

In the **Draft** mode the drawing area can be shifted to the sides by

- Using the scroll bars on the right and bottom
- Using the arrow keys ↑←↓→ on the keyboard

12.3.4. Origin, WCS

In the **2D Design** mode the origin is marked by a circular symbol on the mesh bars at right and bottom. The origin may be moved by entering a new point (Shift Key and left mouse button) and use the button **Set Coordinate Origin**  of the **Advanced** button group.

12.3.5. Simulation domain

The simulation domain is defined by the outermost mesh lines which are displayed at the mesh bars on the right and on the bottom.

Depending on the boundary conditions the simulation domain is marked by lines which are either

- **solid** - **dark red** electric boundary condition
- **solid** - **green** magnetic boundary condition
- **dashed** - **grey** open boundary condition

Hint: Boundary conditions

The boundary conditions can be set in **Simulation Setup**  - **Boundary**

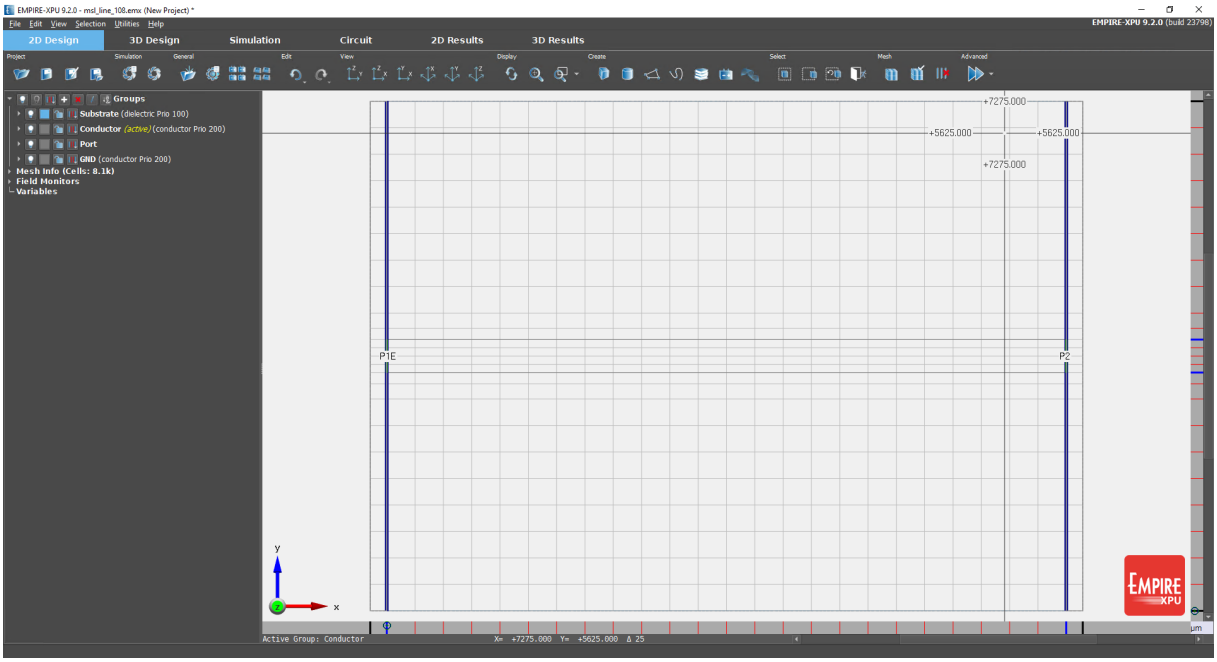


Figure 12.12.: 2D Design mode display showing mesh lines

12.3.6. Bottom line

At the bottom beneath the drawing area there are fields for

- Name and color of current (active) group.
- Number of selected objects and operators.
- Either current cursor and arrow coordinates or current operation.
- Unit

Hint: Active group

Objects are always created in the active group. To activate another group click the right mouse button on one of the group names in the groups list and select **Set Active**

Hint: Operation errors

The field where the number of selections is displayed shows the status of operations by the background color which is green (Status OK) by default. In case of a red background an error has occurred, press Esc to continue.

On the right and at the bottom there are mesh bars displayed, which show the current meshing. By starting from scratch no mesh lines are available yet and only the origin is displayed by small circles ⊕.

Hint: Mesh generation

A grid is generated automatically before starting simulation. It can be created at any time by pressing **Create Mesh**

12.3.7. 2D Outline View

By default, objects are displayed in a wire frame mode showing edges only edges. With a right click the **Outline View** can be changed:

- Outline (wire frame, default)
- Filled faces, overlapping displayed in group order, transparency as set in 3D mode
- XOR mode, Filled faces, overlapping displayed in grey scale

12.3.8. Toggle Groups in 2D Mode

By default, all objects are displayed on top of each other, depending on the group order. With **Ctrl** and **Wheel** the groups can be toggled in order to display single groups, only.

12.4. 3D Results Mode

In order to verify the input, the structure can be rendered and displayed in 3D. Further it is possible to visualize near fields and far field patterns in this mode. When this mode is enabled the group icons change in order to control the visibility, color, or transparency.

12.4.1. Background

By default the background color is black. It can be changed in the Editor Options:

- Editor Options  - Basics - Background Color

12.4.2. Zoom

In the **3D Results** mode the drawing area can be zoomed by

- Using the Top icons 
- Using the mouse wheel
- With **Ctrl + Shift + drag LMB** vertically (left mouse button) a continuous zoom is possible

12.4.3. Pan

In the **3D Results** mode the drawing area can be shifted to the sides by

- Dragging the left or middle mouse button

12.4.4. Rotate

In the **3D Results** mode the structure can be rotated by

- Dragging the right mouse button (or **Ctrl** and left mouse button) the structure is rotated around the horizontal or vertical axis.
- **Ctrl + drag RMB** (right mouse button) the structure is rotated around the orthogonal axis.

Hint

The axis of rotation is the center by default. The center can be set with right click on structure and select **Set Center of Rotation** in the menu.

12.4.5. Display Buttons

Additional Top Buttons:

- **Opposite View** - Display the current backside
- **Cutting tool** - Slice through the structure using the sliders on the left
  - **x** - Shows yz plane in slice window, sliders act in vertical z (horizontal y) direction
  - **y** - Shows xz plane in slice window, sliders act in vertical z (horizontal x) direction
  - **z** - Shows xy plane in slice window, sliders act in vertical y (horizontal x) direction
  - **+/−** - Shows upper or lower (vertical) or left or right (horizontal) part of structure
  - **hv** - Switches between vertical and horizontal slicing planes
- **Axis scaling tool** - Stretch the structure in xyz directions
  - **x-scale** - Acts in x-direction
  - **y-scale** - Acts in y-direction
  - **z-scale** - Acts in z-direction
- **Meshed 3D Structure** - Toggle current scene between discretized and non-discretized objects
- **Export Animation** Save animation as .gif movie
- **Stop Animation** - Stop the animation loop (time, frequency, phase loop)
- **Restart Animation** - Restarts the animation loop (time, frequency, phase loop)
- **Export Image** Save image as .png graphic
- **Mesh Visualization** - Display boundaries and mesh lines relative to the structure
  - **Transparency** - See-trough factor
  - **Mesh x** - Display x mesh lines
  - **Mesh y** - Display y mesh lines
  - **Mesh z** - Display z mesh lines
  - **Boundwidth** - Linewidth of the boundary edges
  - **Meshwidth** - Linewidth of the mesh lines
  - **Linegrey** - darkness of the grey color

- **Enable Opacity** Switch transparency on or off
- **Enable Captions** Switch port legends on or off

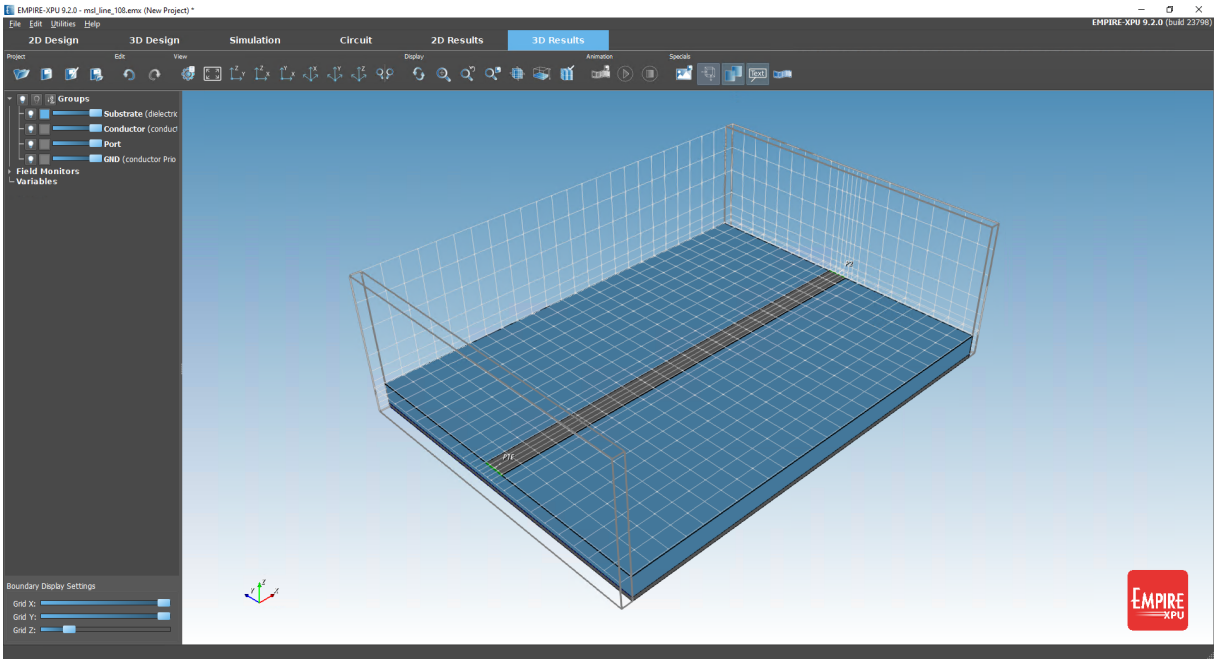


Figure 12.13.: 3D Results mode showing mesh lines

12.4.6. Context Menu

With a right click in the display area more actions are available such as:

- When a field monitor is displayed:
  - **Add Marker** Place a marker at current position
  - **Field Monitor** Edit monitor settings and export field data for current animation scene
  - **Next Animation Frame** Show next step in loop (**Ctrl** + wheel forward)
  - **Previous Animation Frame** Show previous step in loop (**Ctrl** + wheel backward)
- **Set Center of Rotation** Set rotation point at current position
- **View Style** Surface display modes
- **Export Scene** Export display in different formats (.x3d, .vrml, .pov)
- **Export View** Export and save current view as .3dv file
- **Import View** Import view which was exported as .3dv file

Hint

To sweep through the animation loop it is also possible to use **Ctrl** and mouse wheel forward/backward.

12.5. Operation buttons

The top icons (operation buttons Figure 12.14) on the right appear or disappear depending on the current object or operator selections (case sensitive buttons). Their usage may be performed in both forward and backward manner.



Figure 12.14.: EMPIRE XPU™ flexible icon tool bar

Hint: Forward usage

1. Press one of the operation buttons
2. Follow the steps (yellow hints) indicated st the bottom left area of the window.

Hint: Backward usage

1. Select one or more object(s), or enter operator (point/arrow/...)
2. Press one of the operation buttons

While the forward usage is more intuitive the backward usage can be faster and more powerful for the advanced user. (E.g. entering multiple arrows and pressing **Create Box** afterwards generates many boxes instead of only one in the forward usage) A comprehensive list of most available operations can be found in chapter 33

Hint: Context menu

All operations which are currently available can be accessed in a menu by pressing the right mouse button, see Figure 12.15

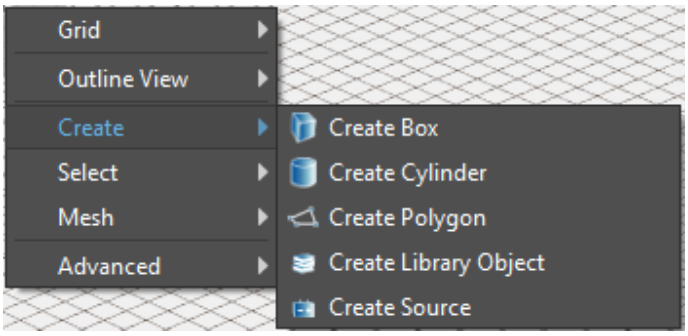


Figure 12.15.: Context menu (right mouse click)

Hint: On-line help

Help is available for most icons and input fields by hovering the mouse over the respective item, see Figure 12.16

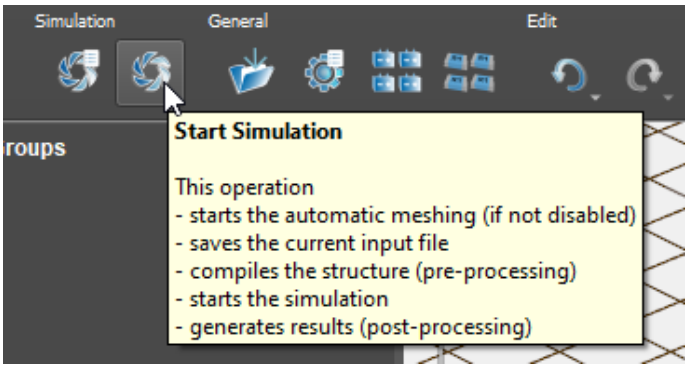


Figure 12.16.: Help on item (hover pointer over icon or input field)

12.6. Side menu / Navigation tree

On the left side of the window there are areas for

- the side menu
- grid and coordiantes
- display status, e.g. progress bar

The side menu 12.17 is used to create or control groups, coordinate systems, field monitors, and variables. To open an item of the side menu click the corresponding sign ▶. To close again click on the sign ▼.

- **Groups** are used for objects with common properties and will be explained in chapter 14. All objects are created in groups and inherit their properties, such as default extrusion height, color, or physical parameters. Components and Sources are listed in their respective groups.
- **Coordinate Systems** List of currently available coordinate systems. The World coordinate system (WCS) is always present and represented by x,y,z Cartesian coordinates. Local Coordinate systems (LCS) can be setup by the user with right click and selecting the ✚ button or with the aid of the context menu in the **3D Design** mode. A coordinate system can be activated by right click and select **Set Active**. The u,v,w tripod represents the origin and orientation of the active LCS
- **Mesh Info** The mesh statistics can be found here. More info is available in section 22.1.
- **Field Monitors** Section for the definition of near field and far field recordings. After simulation the results can be animated and controlled here, too. See Chapter 19.
- **Variables** Both geometrical and physical properties may be defined parametrically. They are entered as character strings, e.g. **xvar**, in object co-ordinates or in most input fields, e.g. object properties. After their range has been defined, these parameters are available in this list and can be controlled here. Variables and equations are explained in chapter 21.

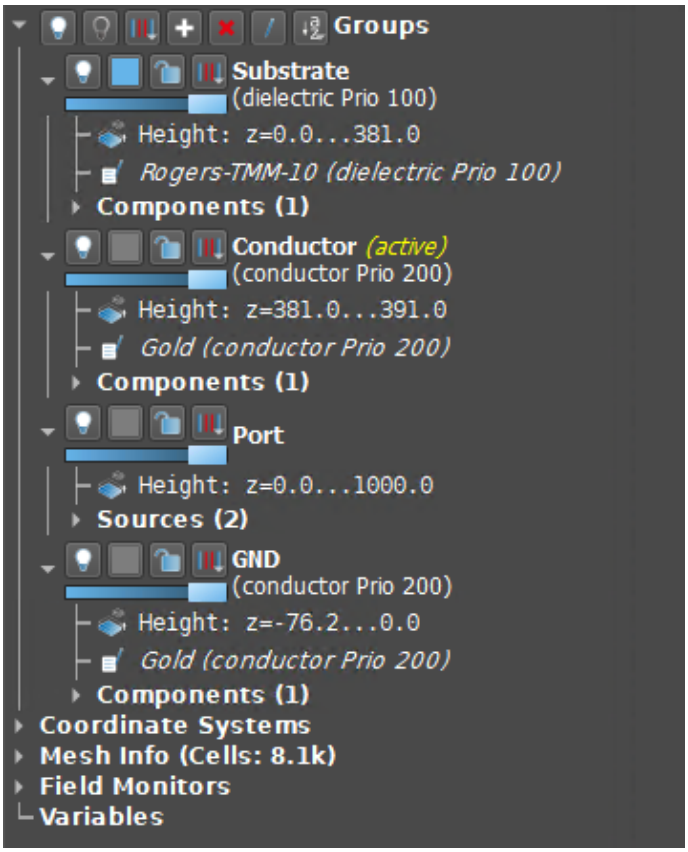


Figure 12.17.: EMPIRE XPU™ navigation tree

13. Operators, Selections and Shortcuts

Many functions in EMPIRE XPU™, like create, select, transform, ... are accessible through operation buttons which are gathered above the drawing area. Depending on the input the arrangement of these buttons change so only useful functions are available.

As input EMPIRE XPU™ regards:

- **Operators**, e.g. points, arrows
- **Selections**, e.g. objects (components or sources), groups, monitors

The number of selected objects are displayed as **Selections** in the left bottom corner.

13.1. Operators

Some operations require the input of an arrow or a point. They can be entered as follows.

Arrows

1. Moving the cursor to the start point of the arrow
2. Press and drag the left mouse button to the end point of the arrow

They are used for e.g. creation of boxes or circles, selection or mirror planes.

Points

1. Moving the cursor to the point
2. Press Shift and the left mouse button

They are used for e.g. creation of polygons, alignments, or rotational axes (together with current grid).

13.2. Object types

For optimized memory management objects are divided in several categories. Objects are listed in **Groups** of the side menu and their category is displayed.

- **Box** 2-point representation, orthogonal to u,v,w coordinates of an LCS
- **Poly** closed polygon with uv-points and constant height in w in an LCS
- **RotPoly** revolved closed polygon with orthogonal rotational axis
- **Surface** arbitrary 3D volume defined by a closed surface or 2D surface (polygonal mesh in STL native format)
- **Library Object** collection of predefined shapes
- **Sources** collection of ports and excitations

Some operations are limited to certain object types. In some cases objects can be converted to other types.

13.3. Select objects

It is often required to select objects. To select one or more objects several methods are available:

- Move the cursor to an object (surface, edge or corner) and press the **left** mouse button
- If objects lay on top of each other
  - **2D Design**: press and drag the **middle** mouse to toggle between objects
  - **3D Design**: press **Shift + right** mouse button to open a select menu which list all objects under the cursor
- Open the **Components** or **Sources** list in a group, and press the left mouse button on the respective object(s). While moving the mouse over the group name its objects are highlighted for identification.
- Press the keys **Ctrl** and **a** to select all (visible) objects
- Press the button **Select overlapping** and enter an arrow which partially or totally covers the object(s)
- Press the button **Select enclosed** and enter an arrow which totally covers the object(s)
- Enter an arrow which does not touch or intersect objects to be selected and press the buttons **Advanced** and **Select outside**
- To select the most recent selected object press **Last selection**
- Recall (**F1, F3, F4**) a previously selected and stored (**Shift F1, Shift F3, Shift F4**) object from the internal clipboard

13.4. Handles

If an object is selected the object is highlighted and small **handles** are displayed on edges (boxes) or corners (polys and solids). The handle which is the nearest to the cursor will be activated (green). Handles can be activated or deactivated (red) with the left mouse button. Multiple handles can be activated by entering an arrow covering multiple edges of the (unselected) object.

- Move and Copy operations act on all handles
- Stretch operations act on selected handles

13.5. Deselect objects

To deselect one or more objects several methods are available:

- Press **Escape** on the keyboard or button to deselect all selections
- Press **Backspace** on the keyboard to deselect the last selected object
- Press **Shift Backspace** on the keyboard to deselect the first selected object
- Move the cursor to a selected object and press the **left** mouse button
- In the **Groups** list, select another group item to deselect a selected component
- Enter an arrow which covers the object(s) and press the button **Deselect overlapping**
- Enter an arrow which totally covers the object(s) and press the button **Deselect enclosed**

13.6. Shortcuts

The following shortcuts are available:

Page up/down	Zoom in/out	
Arrow up/down/left/right	Pan window	
z	Zoom extents or zoom to arrow or object	
w	Zoom window	
Ctrl + e	Zoom extents	
g	Show/hide grid	3D Design
Space	Swap grid plane at cursor position	3D Design
Shift + s	Toggle "Stay on Grid"	3D Design
l	Select last created object	
Ctrl + r	Repeat last action	
Ctrl + Shift + a	Search action	
Ctrl + z	Undo	
Ctrl + Shift + z	Redo	
Ctrl + v	Paste selections from clipboard	
Ctrl + a	Select all visible objects	
Ctrl + s	Save project	
Ctrl + n	New project	
Ctrl + o	Open project	
Ctrl + q	Exit project	
Ctrl + g	Export Gerber file	2D Design
Ctrl + wheel	Next/Previous animation frame	3D Results
F[1,3,4]	Recall selection	

After selection:

Esc	Deselect all	
Backspace	Deselect last	
Shift + Backspace	Deselect first	
Del	Delete selection	
o	Open Object	
Ctrl + c	Copy Selections to clipboard	
Ctrl + x	Move Selections to clipboard	
Ctrl + F[1,3,4]	Append selection	
Shift + F[1,3,4]	Store selection	

After arrow input:

w	Zoom window	
p	Select overlapping	2D Design
e	Select enclosed	2D Design
m	Negate arrow	
+	Concat arrow	
/	Subdivide Arrow	2D Design
*****	Scale Arrow	2D Design

More shortcuts are available for some operations and displayed with balloons while pointing on the respective buttons.

14. Groups

All objects (components and sources) are organized in groups and inherit their group properties (i.e. material, color). Components and sources are listed in their groups and can be selected or accessed for modification.

New objects are always created in an active group. The active group is indicated by a yellow *(active)* behind the group name. In addition the active group is displayed in the left bottom of the editor.

Hint: Group activation

As a shortcut a group can be activated with a middle mouse click on the group name. Alternatively use right click on name and select menu entry **Set Active**

Hint: Group height

In the 2D Design Mode, the group height and direction is used as default extension for boxes, polygons, ports and library elements. During object creation in the 3D Design mode, the group height can be applied with a **long left click** when requested. Using the height is optional and should be adjusted before object creation.

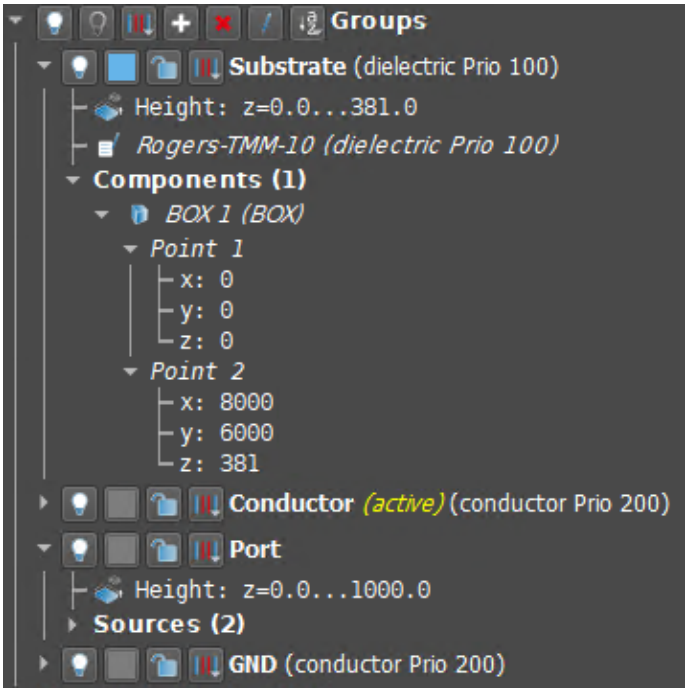


Figure 14.1.: Group menu

14.1. Control for all groups

The top group icons provide control for all groups.



Figure 14.2.: Icons for all groups

- - Switch on all groups
- - Switch off all groups
- Mesh level for all groups (object relevance for mesh creation)
  - - Default mesh level, mesh objects edges and interior
  - - Mesh for objects edges only
  - - No mesh lines for objects
  - - Mesh object edges and fine interior
  - - Ignore edges, interior only
  - - Mesh gaps, only
  - - Mesh object bounding box only
- - Create a new group with default properties
- - Delete unused (empty) groups
- - Start group tree editor, see Fig. 14.3
- Group sort
  - - Sort groups alphabetically
  - - Sort groups alphabetically in reverse order
  - - Sort groups according to increasing group height
  - - Sort groups according to decreasing group height
  - - Sort groups according to increasing object surfaces
  - - Sort groups according to decreasing object surfaces
  - - Sort groups according to increasing priority
  - - Sort groups according to decreasing priority

Right click on **Groups** opens a context menu with additional functions such as **Edit Group Tree**. Here, all group settings can be accessed quickly. In addition a complete group tree can be exported and re-used in another project by importing the tree.

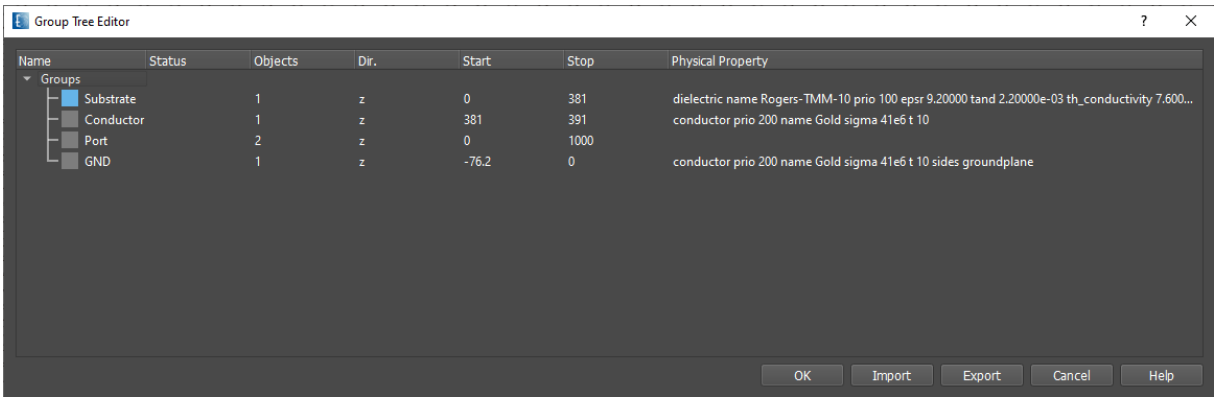


Figure 14.3.: Group Tree Editor

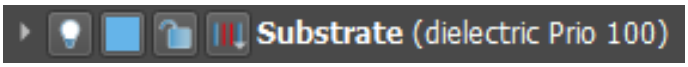


Figure 14.4.: Icons for single group

14.2. Control for single groups

The following icons are available in each group.

- - Switch group on/off
- - Select group color
- - Lock/unlock group to prevent its objects from selection
- Group mesh level (object relevance for mesh creation)
  - - Default mesh level, mesh objects edges and interior
  - - Mesh objects edges only
  - - No meshing for objects in this group
  - - Mesh object edges and fine interior
  - - Ignore edges, interior only
  - - Mesh gaps, only
  - - Mesh object bounding box only
- - Transparency slider for 3D modes

Right click on the respective group name opens a context menu with additional functions such as **Edit Name**, **Add Property**, **Set Active**, **Select Group's Objects**, etc. After object selection some more functions are available e.g.**Copy selected objects to group**.

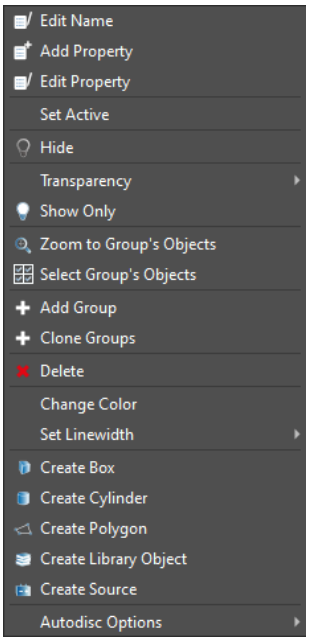


Figure 14.5.: Group menu

Hint: Set Properties

When a new group is created, the default property **conductor** (PEC) is set. If the property should be changed double click on the button **Property** or use **Edit Property** in the group context menu. Properties are explained in chapter 18

Hint: Group name

By default the group name is set to a number, e.g. `Group_004`. It can be edited with a right mouse click on the group name and select **Edit Name**.

Hint: Sub Groups

Groups can be organized into sub groups with the special character `|` in the group name. Use, e.g. `Antenna|Feed` and `Antenna|Dish` to create a group **Antenna** with sub groups **Feed** and **Dish**. The sub groups inherit the property of the parent group but can also be assigned individually.

Hint: Drag & Drop

Drag and drop operations can be used:

- Drag group name and drop between groups to place a group to another position
- Drag group name and drop on another group name to move it into another group as a sub group
- Drag height and drop on another group to copy the height from one group to another
- Drag property and drop on another group to copy the property from one group to another
- Drag components and drop on another group to move the components from one group to another

15. Object creation

This chapter gives an overview on how to create objects in EMPIRE XPU™. Building more complex geometries, e.g. shaping, drilling, merging, will be described in chapter 16.

Hint: Object creation

New objects are created in the active group and inherit the group **property** (conductor, dielectric, ...). All mesh cells which are covered by the object are assigned with the group property.

In the **3D Design** mode objects can be defined in arbitrary coordinate systems (WCS, LCS). Before input the coordinate system and grid should be selected (e.g. set u=100). By checkmarking the **Stay on Grid** switch, the cursor stays on the grid plane, otherwise it snaps on visible object surfaces. The height is defined normal to the grid.

In the **2D Design** objects are always defined in the World Coordinate System (WCS). Some objects (Boxes, Polygons, 2D Extrusions, 2D Layout imports...) make use of the group **height** during creation. Other objects (Ports, 3D shapes, ...) use the group **start height** for the definition of the perpendicular position. 3D imports ignore the height information.

If the group height should be used, it is advisable to set the group height accordingly before object creation.

During input zoom, rotate and pan can be used to access odd coordinates. After input, object coordinates and parameters can also be set in an editor window or in the group list, e.g. to adjust values which could not be accessed with the cursor.

Objects may also be defined with parameters, e.g. for variation and optimization purposes.

Hint: Parametric objects

By entering a variable name, e.g. **anyname** instead of a number for a co-ordinate the value can be controlled in the **Variables** list in the side menu. If the variable is new, its range, current value and step size have to be entered.

There are mainly 3 approaches on how to create objects:

- **Forward usage:** Clicking a button in the **Create** menu and entering values or operators afterwards
- **Backward usage:** Entering operators (points, arrows) and pressing the create object button afterwards
- **Import:** Use the import button and load specific 2D or 3D data files.

15.1. Basic Objects

Basic objects are Boxes, Polygons, Line and Rotational Polygons. Their edges or corners are represented by handles (visible after selection) which can be stretched for modifications. After creation these objects are available in the **Groups** list on the left.

15.1.1. Box

A box is defined by 6 numbers which represent two opposite points in the 3D space.

3D Design

Choose the grid for the cross section which is normal to the height. Optionally, adjust the group height if it should be used for creation.

Forward Usage

1. Click button **Create Box** 
2. Left click to enter first corner
3. Left click to enter cross section
4. Left click on height position or long click to use group height

After creation an editor window is displayed and values can be adjusted if necessary.

Backward usage

1. Draw an arrow (drag left mouse button) representing the box cross section
2. Click button **Create Box** 
3. Select height in editor (from group or custom and enter values)

2D Design

Choose the view for the cross section which is normal to the height Optionally, adjust the group height if it should be used for creation.

Forward Usage

1. Click button **Create Box** 
2. Enter coordinate values of the box or right click on **Point - Pick Coordinate** for cursor input
3. Select height in editor (from group or custom and enter values)

Backward usage

1. Draw an arrow (drag left mouse button) in the drawing plane representing the box cross section
2. Click button **Create Box** 
3. Select height in editor (from group or custom and enter values)

15.1.2. Circular Cylinder

In EMPIRE XPU™ a circle is a 1-point polygon with a radius > 0. Together with the height coordinates a circular cylinder is defined.

3D Design

Choose the grid for the cross section which is normal to the height. Optionally, adjust the group height if it should be used for creation.

Forward usage

1. Click the icon **Create Cylinder** 
2. Left click at midpoint of the circle
3. Left click at any point on the circumference of the circle
4. Left click at height position or long click to use group height

After creation an editor window is displayed and values can be adjusted if necessary.

Backward usage

1. Enter an arrow in the drawing area from midpoint to any point on the circle (drag left mouse button).
2. Click the icon **Create Cylinder** 
3. Select height (from group or custom and enter values)

2D Design

Choose the view for the cross section which is normal to the height. Optionally, adjust the group height if it should be used for creation.

Forward usage

1. Click the icon **Create Cylinder** 
2. Enter x and y values of the midpoint or right click on **Origin - Pick Coordinate** for cursor input
3. Enter the radius r
4. Select height (from group or custom and enter values)

Backward usage

1. Enter an arrow in the drawing area from midpoint to any point on the circle (drag left mouse button).
2. Press the icon **Create Cylinder** 
3. Select height (from group or custom and enter values)

15.1.3. Polygon

An N-point polygon is defined by (2N+2) numbers which represent the N-point cross section and two height co-ordinates. A polygon is closed by connecting first and last point. These objects are mainly used in planar layouts. They can be created as follows.

3D Design

Choose the grid for the cross section which is normal to the height. Optionally, adjust the group height if it should be used for creation.

Forward Usage

1. Click button **Create Poly** 
2. Left click to enter first corner
3. Left click(s) to enter next corner(s)
4. Left click on first corner to close the polygon
5. Left click at height position or long click to use group height

After creation an editor window is displayed and values can be adjusted if necessary.

Backward usage

1. Enter a set of points in the drawing area (Shift and left mouse button)
2. Click button **Create Poly** 
3. Select height (from group or custom and enter values)

2D Design

Choose the view for the cross section which is normal to the height. Optionally, adjust the group height if it should be used for creation.

Forward Usage

1. Click button **Create Poly** 
2. Enter coordinate values of the corners
3. Right click on **Point** to insert  or delete  a point
4. Right click on **Point** and **Pick Coordinate**  to acquire a coordinate with cursor
5. Select height (from group or custom and enter values)

Backward usage

1. Enter a set of points in the drawing area (Shift and left mouse button)
2. Click button **Create Poly** 
3. Select height (from group or custom and enter values)

Remarks

- The sequence of points entered determines which points will be connected with lines
- The last point and first point will be connected to close the polygon
- The latest entered point(s) can be deleted by pressing the **Backspace** key

Hint: Holes in Polygons

Holes in Polygons are created by subtracting small from large polygons, e.g. during import. They are defined by a cut connecting inside and outside (two connection lines with zero spacing, represented by 4 points in the component point list). The cut angle can be adjusted in **Editor Options - Compnents - Internal Line Angle**

15.1.4. Strip

A strip is defined by a set of points which represent the corners, the strip width and the strip height. The corners may be rounded.

3D Design

Choose the grid for the cross section which is normal to the height. Optionally, adjust the group height if it should be used for creation.

Forward Usage

1. Click button **Create Strip**
2. Left click to enter first corner
3. Left click(s) to enter next corner(s)
4. Long left click to finish input
5. Left click at height position or long click to use group height
6. In Editor Window click **Edit Settings** to adjust strip width

Backward usage

1. Enter a set of points in the drawing area (Shift and left mouse button)
2. Click button **Create Strip**
3. In Editor Window click **Edit Settings** to adjust strip width
4. Select height (from group or custom and enter values)

2D Design

Choose the view for the cross section which is normal to the height. Optionally, adjust the group height if it should be used for creation.

Forward Usage

1. Click button **Create Strip**
2. Enter coordinate values of the corners
3. Right click on **Point** to insert or delete a point
4. Right click on **Point** and **Pick Coordinate** to acquire a coordinate with cursor
5. Select height (from group or custom and enter values)
6. Click **Edit Settings** to adjust strip width

Backward usage

1. Enter a set of points in the drawing area (Shift and left mouse button)
2. Click button **Create Strip**
3. In Editor Window click **Edit Settings** to adjust strip width
4. Select height (from group or custom and enter values)

15.2. Object Library

More complex objects can be found in the **Object Library** which is available by clicking **Create Library Object**. They are defined by point(s), height and parameters. The points (P1, P2, ...) are used for positioning and are displayed as handles when selected. The height is used to define the z (or w) position, extension or focal point.

Similar to the basic objects the creation can be carried out in forward or backward mode:

Forward Usage

- Click **Create Library Object**
- Select object
- Enter point(s) and height(s)
- Adjust Parameters in **Edit Settings** of the editor window

Backward Usage

- Enter point(s) and height(s)
- Click **Create Library Object**
- Select object
- Adjust Parameters in **Edit Settings** of the editor window

3D Design

After selecting an object in the library, coordinates are requested by cursor input as displayed in the guide at the left bottom. Depending on the shape, either

- center, radius, and height e.g. sphere, helix
- rectangle corners and height, e.g. ellipsoid, pyramid
- a set of points with their heights, e.g. wire, polygon

have to be entered on the current WCS or LCS grid.

During input zoom, pan and rotate view may be used for positioning. After creation, an editor window is opened and all values and can be edited.

2D Design

After selecting the object will be created immediately in the current view plane with default parameters and coordinates which values can be edited in the editor window. Cursor input is supported by right click on the point list .

Most of the objects can be scaled, copied, moved, rotated or mirrored. If the operation is not possible or if these objects should be modified by Boolean operations, they have to be converted first to solids or polygons by the **Explode** operation and optional variables will be replaced by their current values.

15.2.1. 3D Solid

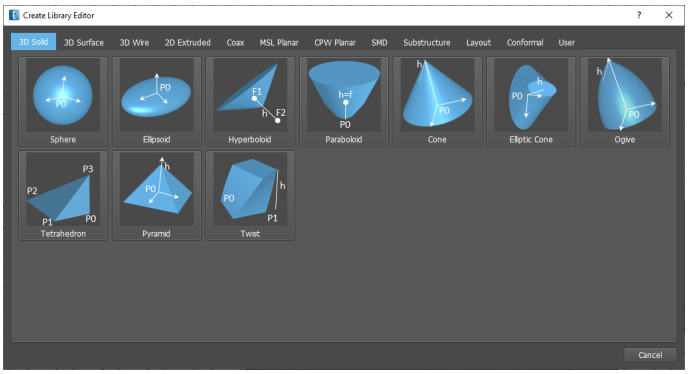


Figure 15.1.: View of 3D Solid Library

Will be converted to Solids after explosion.

- **Sphere** The point and the height define the midpoint of the sphere. Parameters are used to define radius, cut angle and resolution.
- **Ellipsoid** The point and the height define the midpoint of the ellipsoid. Parameters are used to define major axes, cutting planes and resolution
- **Hyperboloid** Here, the height defines the two foci of the rotational one-shell hyperboloid. Parameters are used for major axes, diameter and resolution. The sign of the major axis determines which one of the shells is used.
- **Paraboloid** The height defines tip and focus of the rotational paraboloid. Parameters are used for diameter and resolution.
- **Cone** The point and the height define the midpoint and the perpendicular extension of the cone. Parameters are used to define bottom and top diameter and resolution
- **Ogive** The point and the height define the midpoint and the perpendicular extension of the ogive. Parameters are used to define diameter and resolution
- **Tetrahedron** The four points define the coordinates in the workspace. The parameters define the perpendicular coordinates of the tetrahedron corners.
- **Pyramid** The point and the height define the midpoint and the perpendicular extension of the pyramid. Parameters are used to define the side lengths
- **Twist** Two points represent the cross section of the bottom rectangle. The height defines the top with a twist angle defined in the parameters.

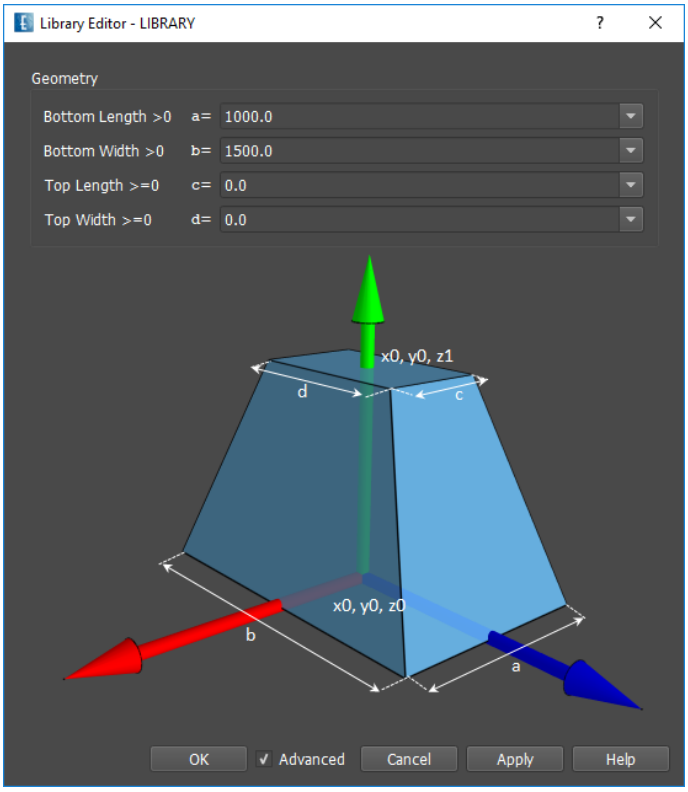


Figure 15.2.: Parameters of the pyramid

15.2.2. 3D Surface

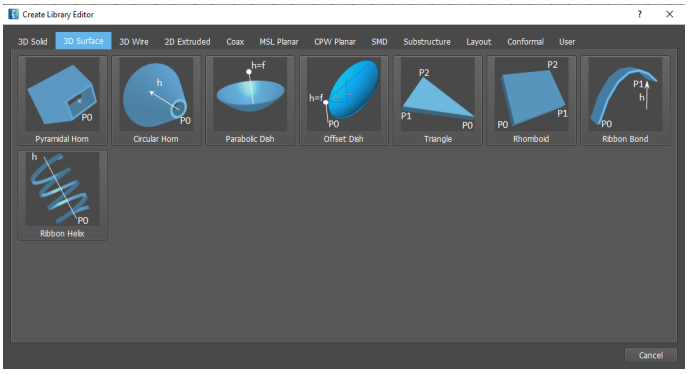


Figure 15.3.: View of 3D Surface Library

Will be converted to Solids after explosion.

- **Pyramidal Horn** The point defines the center of the horn and the height defines the perpendicular extension. In the parameters the bottom and top length as well as the thickness can be set.
- **Circular Horn** The point defines the center of the horn and the height defines the perpendicular extension. In the parameters the bottom and top diameter as well as the thickness can be set. The resolution describes the angle step used for the revolution.
- **Parabolic Dish** The point defines the center of the dish and the height defines the focal length. In the parameters the diameter as well as the thickness can be set. The angular resolution describes the angle step used for the revolution. The horizontal resolution defines the step size in units.
- **Offset Dish** The point defines the center of the (unrotated, zero-offset) dish and the height defines the focal length. In the parameters the diameter as well as the thickness can be set. The angular resolution describes the angle step used for the revolution. The horizontal resolution defines the step size in units. The illumination direction defines the direction of the plane wave (for zero rotation angle) and the offset direction has to be set perpendicular to it. The offset is the distance between center point and dish plus half diameter of the unrotated reflector. A rotation angle can be applied which rotates the dish around the focal point. If the rotation angle is set to **auto** an angle is chosen that the focal line is parallel to the illumination direction. With annotation characteristic lines and angles can be displayed, which show e.g. focal point and plane wave direction.
- **Triangle** The 3 points define the corner of the triangle. In the parameters the thickness can be set. If the thickness is entered negative, the extrusion is done in the opposite direction.
- **Rhomboid** The three points define the cross section of the object. The thickness can be set in the parameters.
- **Ribbon Bond** The 2 points together with the height define start and end coordinates of this element. In the parameters the width and thickness as well as the slope angle can be set. The number of support points define the resolution of the piecewise linear shape approximation. A horizontal at the ribbon ends cut can be defined.
- **Ribbon Helix** The point defines the center of the helix and the height defines the helix height. In the parameters the width and thickness can be set as well as the diameter, number of turns, start angle and circle resolution. The number of turns may be fractional.

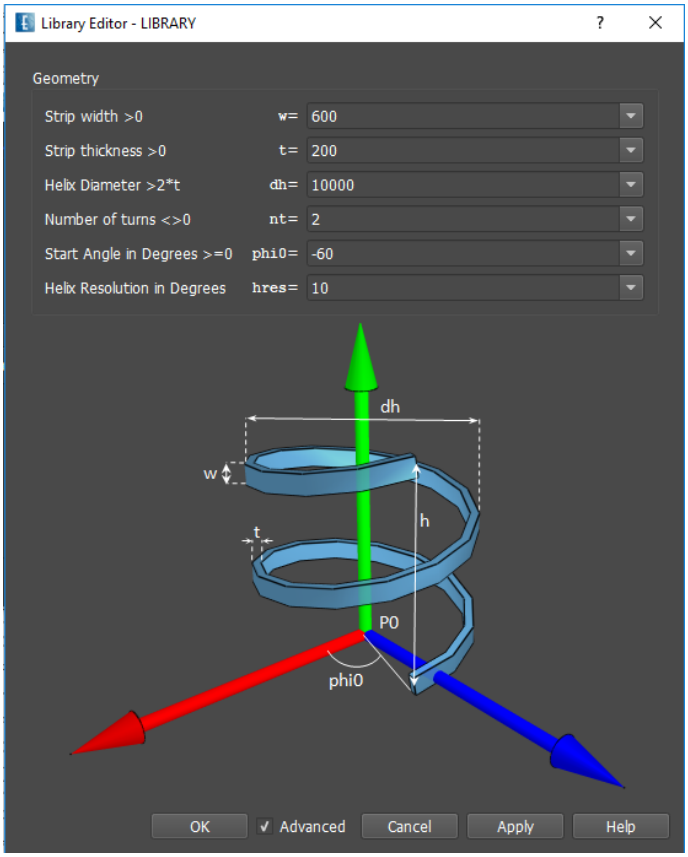


Figure 15.4.: Parameters of the ribbon helix

15.2.3. 3D Wire

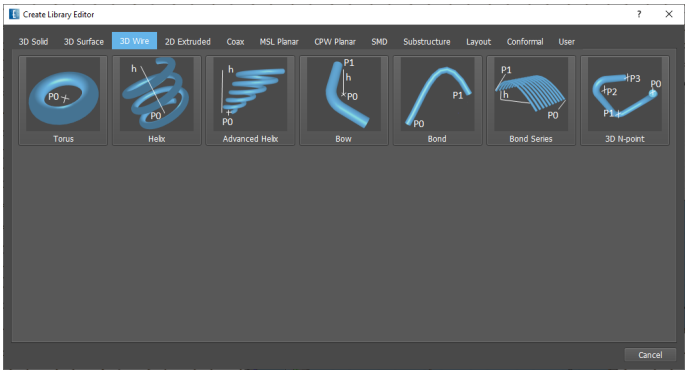


Figure 15.5.: View of 3D Wire Library

Will be converted to Solids after explosion.

- **Torus** The point and the height define the midpoint of the torus. Parameters are used to define wire and ring diameter, start and stop angle and resolution.
- **Helix** The point defines the midpoint of the helix. Parameters are used to define wire and helix diameter, start angle, resolution and number of turns which may be fractional.
- **Advanced Helix** This object is defined by a wire (diameter dc) wrapped around a center line (defined by shift angle phis, displacement shift and height h) with normal vector perpendicular to h. Helix diameters at bottom and top (dh0 and dh1) may be different. Number of turns nt may be fractional, start angle phi0 and shift angle phis are defined in the plane orthogonal to h
- **Bow** Two straight wires with a circular rounding in a plane. One point defines the start of the bow and the other point defines the center of the arc. Diameter, end point and bend radius are set in the Parameters.
- **Bond Wire** Point 0 and minimum height define the start point and Point 1 and maximum height define the end point of this object. Parameters are used to enter the wire diameter slope angles and resolution. Optionally, a horizontal cut can be applied so the object does not intersect with pads.
- **Bond Wire Series** Point 0 and minimum height define the start point and Point 1 and maximum height define the end point of this object. Parameters are used to enter the wire diameter and number of wires, total width of the series, slope angles and resolution.
- **3D N-point Wire** Multiple straight wires with circular rounding. The points define start, end or the intersection of the straight wires. Diameter, and bend radius are set in the Parameters. The minimum bend radius is defined by the wire diameter.

15.2.4. 2D Extruded

After explosion these objects will be converted to polygons.

- **Regular Polygon** Point 0 defines the center of the polygon while point 1 sets one of the corners. In the Parameters the number of sides or corners can be specified.
- **Ellipse** The point defines the midpoint of the ellipse. Parameters are used to define the major axes and resolution

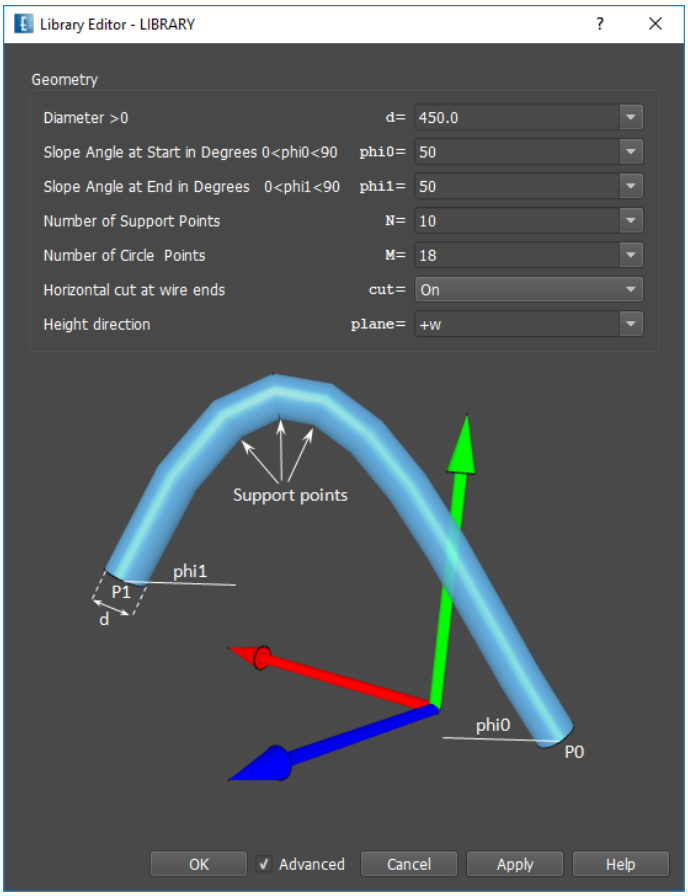


Figure 15.6.: Parameters of the bond wire

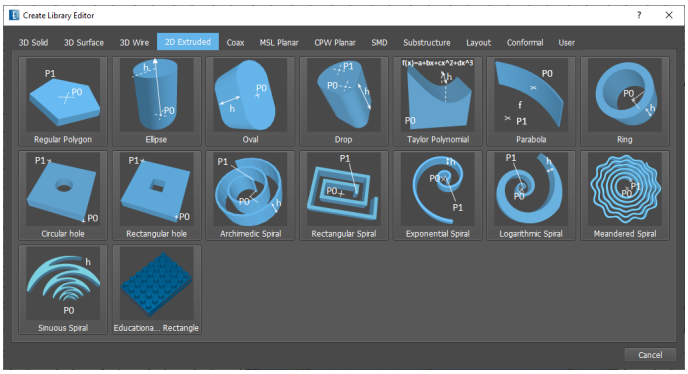


Figure 15.7.: View of 2D Extruded Library

- **Oval** The points define the midpoint of the two half-circles. Parameters are used to define the width and resolution
- **Drop** The drop is an oval with two different radii. The two points define the centers of the rounds. In the parameters the associated radii and a resolution can be set.
- **Taylor Polynomial** The points define start and end point of the polynomial. Parameters are used to define the coefficients and the number of support points
- **Parabola** Point 1 defines the center of the parabola and point 2 the focus of the parabola. Parameters are used to define the diameter, thickness and the number of support points
- **Ring** The point defines the midpoint of the ring Parameters are used to define inner and outer diameter, start and stop angle and resolution
- **Circular hole** The two points define the rectangle. Parameters are used to determine diameter, offsets from center and resolution.
- **Rectangular hole** The two points define the outer rectangle. Parameters are used to determine inner rectangle, offsets from center and resolution.
- **Archimedic Spiral** Point 0 defines the midpoint Point 1 the begin of the archimedic spiral. Parameters are used to define strip and gap width as well as resolution and number of turns which may be fractional.
- **Rectangular Spiral** The points define midpoint and first corner of the rectangular spiral Parameters are used to define vertical and horizontal strip and gap width and the number of turns
- **Exponential Spiral** The points define midpoint and the begin of this spiral Parameters are used to define line width, growth ratio and the number of turns
- **Logarithmic Spiral** The points define midpoint and the begin of this spiral Parameters are used to define angle, growth ratio and the number of turns
- **Meandered Spiral** The points define midpoint and the begin of this spiral Parameters are used to define strip width, gap, growth ratio, number of turns and meander definition
- **Sinuous Spiral** The point define midpoint of this spiral. Parameters are used to define radius, ratio, angles and the number of cells
- **Educational Brick Rectangle** The point define midpoint of the brick. Parameters are used to define grid spacing and offset as well as brick size and direction.

15.2.5. Coax

These elements are compound objects consisssting of inner and outer conductors and dielectric intermdeiate. They will be converted into single parts after explosion.

- **2-Point-Coax** The points define start and end point of the line. The Parameters are used to define the diameters, permittivity and circle resolution.
- **N-Point-Coax** The points define start and end and intermediate points of the line. The Parameters are used to define the diameters, permittivity and circle resolution.

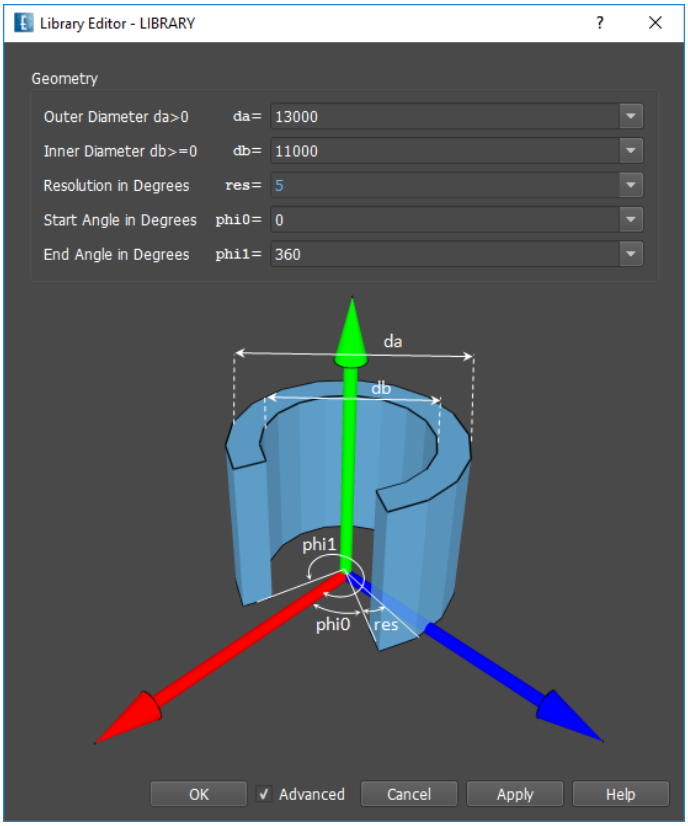


Figure 15.8.: Parameters of the ring

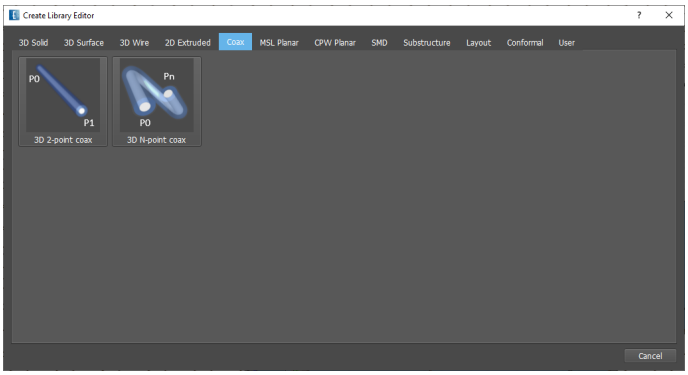


Figure 15.9.: View of Coaxial Library

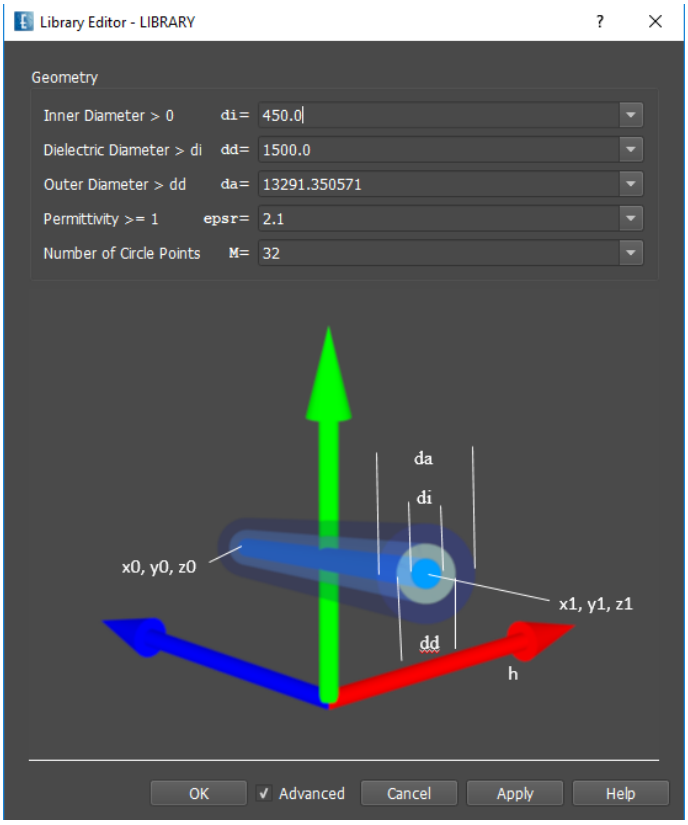


Figure 15.10.: Parameters of the 2-point coaxial element

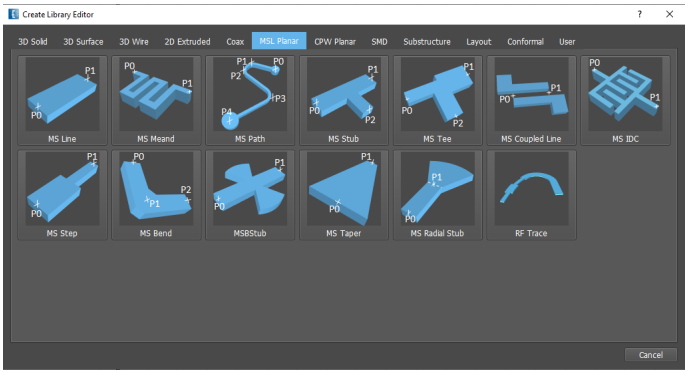


Figure 15.11.: View of MSL Planar Library

15.2.6. MSL Planar

Will be converted to Polygons after explosion.

- **MS Line** The points define start and end point of the line. The Parameter is used to define the line width.
- **MS Meand** The points define start and end point of the line. The Parameter is used to define the line width.
- **MSStub** The points define start and end point as well as the stub position and length. The Parameters are used to define the width of the line and the stub.
- **MS Path** The points define the corners of the path. By default 3 points are defined which can be edited with coordinates or mouse input. More points can be added by using the up/down buttons. A radius may be set at each corner. In the Parameters the width and the resolution can be set. Optionally a default arc radius may be set which applies to all corners where the individual radius is set to 0.
- **MS Tee** The points define the location of the tee line ends. The Parameters are used to define the width of the lines.
- **MS Coupled Line**. The points define the location of the start and end point of the coupling section. The Parameters are used to define the gap, length and width of the lines.
- **MS IDC** Interdigital capacitor. The points define the location of the line ends. The Parameters are used to define the gap and width and number of fingers.
- **MS Step** The points define start and end point as well as the stub position and length. The Parameters are used to define the width of the line and the stub.
- **MS Bend** The points define start and end point of the step. The Parameters are used to define the width of the lines and the position of the step.
- **MS Bstub** The points define start and end point of the butterfly stub. The Parameters are used to define the width of the line, angle of the circular sector, intersection depth, outer radius and the position of the stub.
- **MS Taper** The points define start and end point of the line. The Parameters are used to define the line width at start and end of the taper.
- **MS Radial Stub** The points define start and end point of the line. The Parameters are used to define the line width, radius and angle of the radial stub.
- **RF Trace** This element defines a tapered line with arbitrary number of corner points.In the Parameters taper length and start and end width of each section can be set. Roundings are applied automatically depending on the chosen parameters.

15.2.7. CPW Planar

Will be converted to Polygons after explosion.

- **CPW Line** The points define start and end point of the line. The Parameters are used to define the width, slot and ground width.
- **CPW Taper** The points define start and end point of the line. The Parameters are used to define the width, slot and ground width.
- **CPW Short** The points define start and end point center line. The Parameters are used to define the width, slot and ground width of the line and short length.

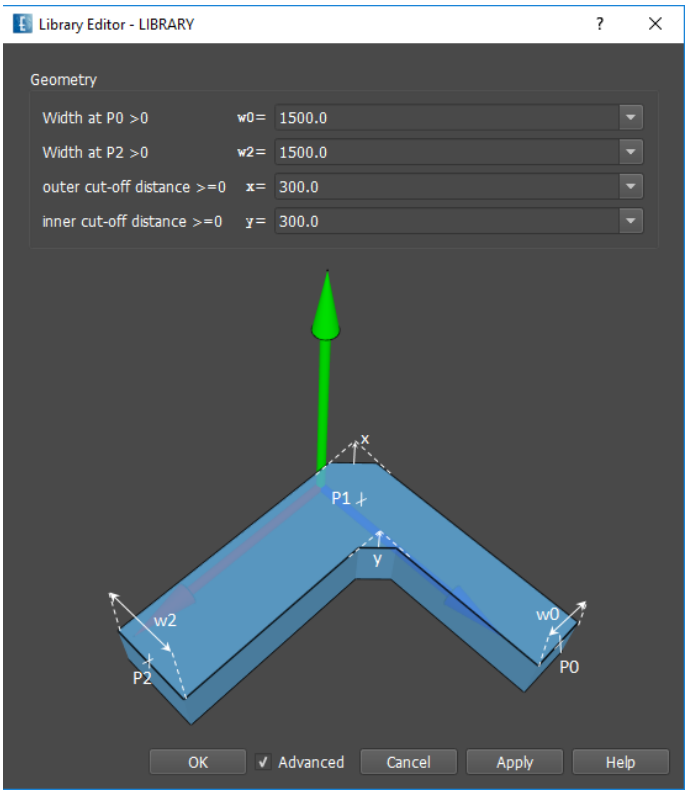


Figure 15.12.: Parameters of the MSL Bend

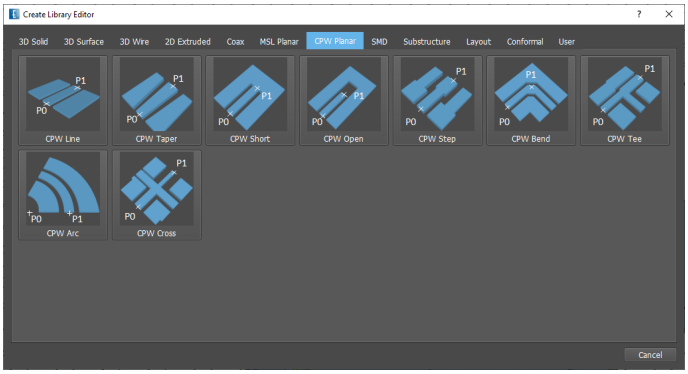


Figure 15.13.: View of CPW Planar Library

- **CPW Open** The points define start and end point center line. The Parameters are used to define the width, slot and ground width of the line and open gap.
- **CPW Step** The points define start and end point. The Parameters are used to define the width, slot and ground width of the line at start and end.
- **CPW Bend** The points define start and mid point of the bend. The Parameters are used to define the width, slot and ground width of the lines and the relative position of the step.
- **CPW Tee** The points define the location of the tee line ends. The Parameters are used to define the width, slot and ground width of the lines.
- **CPW Arc** The points define the center and the start point of the circular arc. The Parameters are used to define the width, slot, ground width and angle.
- **CPW Cross** The points define the location of the start and end point of the cross section. The Parameters are used to define the width, slot and ground width of the lines.

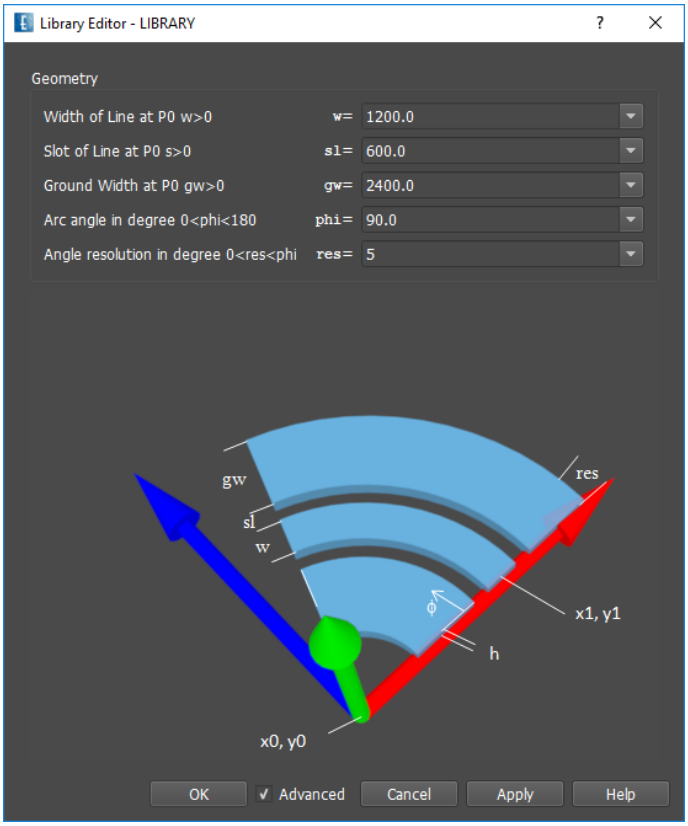


Figure 15.14.: Parameters of the circular CPW arc

15.2.8. SMDs

SMDs are 3D compound objects containing caps, body, pads and distributed circuit element. The perpendicular position of these objects are determined by the group height (2D Design) or grid (3D Design). After explosion, the SMDs are converted to boxes and polygons and grouped in sub folders of group **SMD**. SMD parts of new SMD objects can be defined with group properites. For compatibility reasons legacy SMD objects are still available.

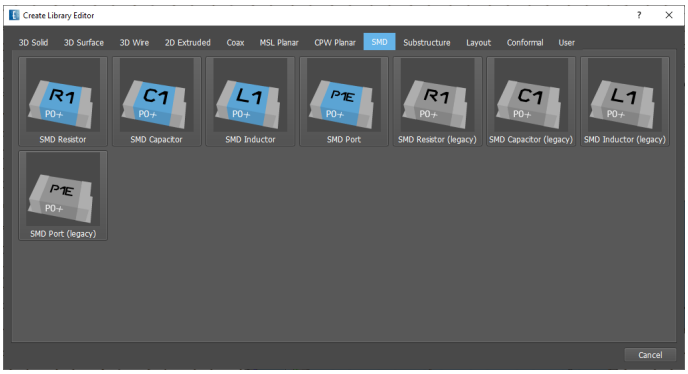


Figure 15.15.: View of SMD Library

- **SMD Resistor** The point defines the midpoint of the SMD. The resistance can be set in Parameters.
- **SMD Capacitor** The point defines the midpoint of the SMD. The capacitancecan be set in Parameters.
- **SMD Inductor** The point defines the midpoint of the SMD. The inductancecan be set in Parameters.
- **SMD Port** The point defines the midpoint of the SMD. The port properties can be set in Parameters.

Further parameters are:

- **SMD size** List of SMD standard sizes
- **Orientation** Horizontal (h) or vertical (v). With the exception of **SMD Port** an angle can be specified for arbitrary orientation
- **Metal thickness** Thickness of the pads
- **Label** Textstring
- **Label size factor** can be used if string is too long
- **Upside Down** for backside mounting
- **Create Pads** can be switched off
- **Interior position** Position of the distributed circuit element
- **Model Type** Type of the circuit element
  - **Full** Complete SMD geometry
  - **Flat** Model in the pad plane
  - **Wire** Simplified wire model
  - **concentrated** lumped circuit element

An SMD Setup Wizard is available for a quick access to the SMD parameters if many SMD elements have been defined.

15.2.9. Substructure

Substructures may be used to import contents of an EMPIRE XPU™ input file into a current project.

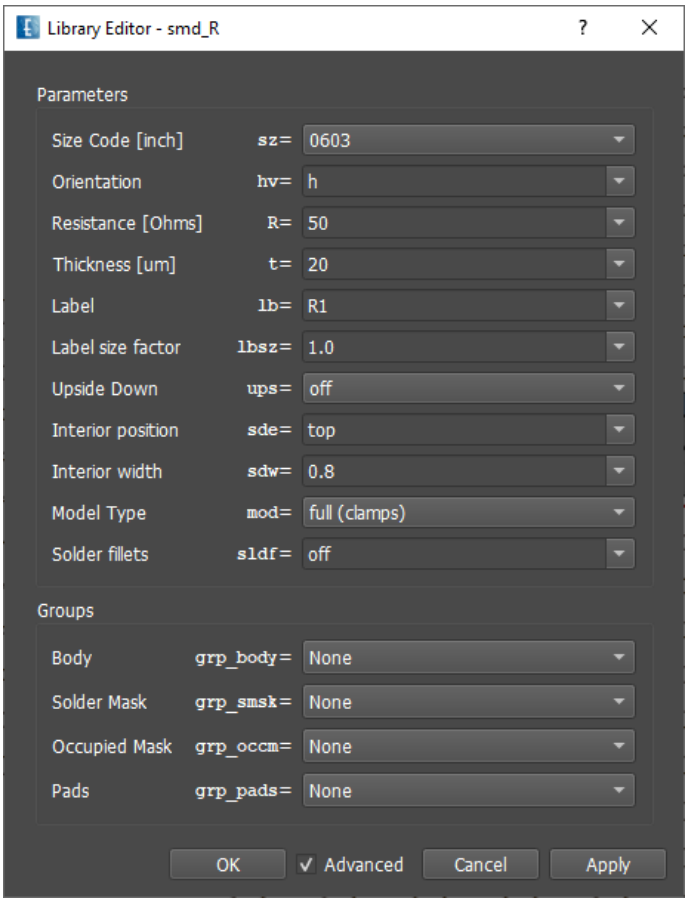


Figure 15.16.: Parameters of the SMD element

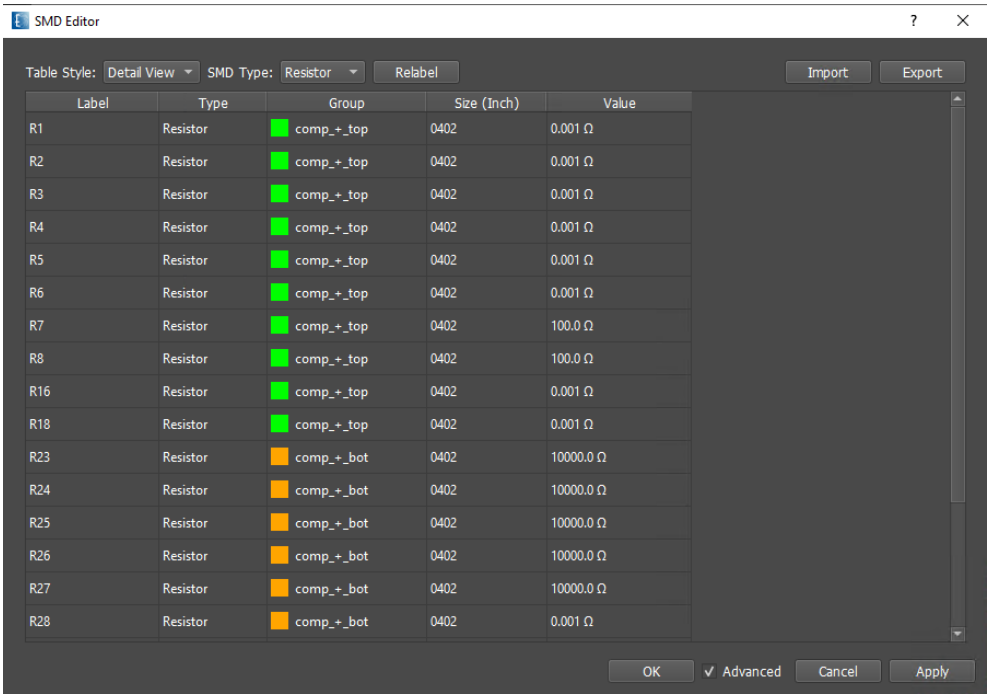


Figure 15.17.: SMD Setup Wizard

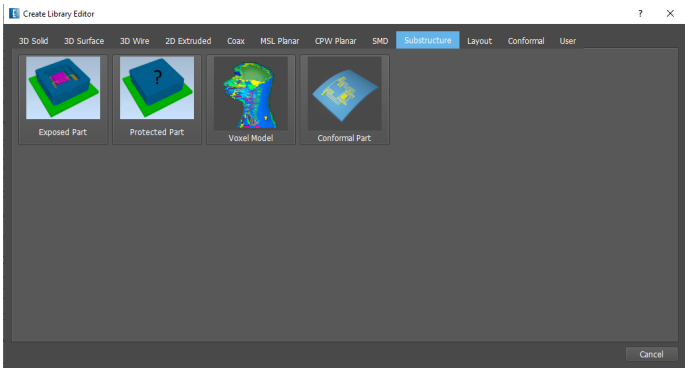


Figure 15.18.: View of Substructure Library

Exposed Part

These substructures are created from EMPIRE XPU™ legacy input files ending **.gym** which have to be located in the current project folder. Their geometry will be inserted into the current drawing and transformations can be applied. Changing the geometry of the source file will be recognized in the current drawing.

- **Substructure** This element loads a source file into the current drawing and applies transformations. The point and the height defines the origin of the substructure in the current drawing. Meaning of the parameters:
  - **General** Here, the file name of the source file has to be specified and a reference point can be entered which refers to the coordinates of the source file geometry
  - **Transformation** Here, a scale factor can be entered to enlarge or decrease the size of the source geometry. Further a mirror can be switched on to create an opposite geometry. A rotation angle can also be specified and is applied to the insertion point P0.
  - **Options** By default the scaling is limited to the transverse direction. If it should be applied to the height direction, too, it can be set here.

Protected Part

This substructure is created from exporting a file **File - Export - Empire - Read Protected Simulation File** ending **.aprot** see 17.2.5. This file and if generated its associated encryption key file **.akey** have to be located in the current project folder. The geometry, electric property, ports and mesh will be inserted into the current drawing without displaying details. A bounding box will be used to display the volume of the protected part. Outside the bounding box surrounding objects may be entered. The meshing inside the bounding box will be maintained. The contents of the read protected file can not be edited. Simulation settings (e.g. frequency range, structure type, boundary conditions) are not contained and have to be defined in the current project.

Hint

Field monitors may not be defined inside the area which is defined by the protected part, i.e. a **Complete EM Field** monitor can not be defined, only partial EM fields.

- The file name **.aprot** has to be specified in **Parameters**
- The origin of the protected part can be specified with the coordinates x,y,z

Hint: Application Note

More information is available in Chapter 29.2.18

Voxel Model

These substructures are created from EMPIRE XPU™ voxel input files ending **.vxx** which have to be located in the current project folder. The model will be inserted into the current drawing with an anchor defined by the **Origin** of the library element.

With **Edit Settings** all voxel properties can be set as defined in Chapter 18.6.1.

Conformal Part

The conformal mapping is created from an EMPIRE XPU™ legacy input file ending **.gym** which has to be located in the current project folder. Its geometry (usually a planar layout) will be projected on a selected surface. Changing the geometry of the source file will be recognized in the current drawing.

- The mapped structure has to be smaller than the surface
- The projection direction and orientation is defined by a local coordinate system which maps the x,y,z coordinates of the source file to the u,v,w direction of the LCS.
- The element point specifies the origin on the selected surface
- The source file origin is mapped to the element point origin unless the reference point is set differently.

Proposed usage (assuming xy-planar layout in draft.gym file)

1. Create an LCS on the target object with **w** normal to surface
2. Adjust direction of **u,v** corresponding to **x,y** of source file
3. Create Library Object - Substructure - Conformal Part
4. Set element point origin, e.g. click on LCS origin

Menu items:

- **Mapping**
  - **Map resolution** The resolution of the conformal geometry can be specified in units or percent of the bounding box.
  - **Local Coordinate System** The surface orientation at Point 0 can be set by an LCS.
  - **Algorithm** used for mapping source object on surface
  - **Options** If set to **keep height** the thickness is not deformed by the projection
- **Substructure**
  - **Filename** Here, the file name of the source file has to be specified
  - **Reference** a point can be entered which refers to the coordinates of the source file geometry.
  - **Object resolution** of the source geometry
  - **Max Object # Triangles** upper limit for the source geometry subdivision

15.2.10. Layout

A collection of useful objects for layout purposes.

- **Measure**, Insert a dimensioning line. The first 2 points define the distance and the third point the label location
- **Angle**, 2D Design, only. Insert an angle dimensioning
- **Label** The font size and orientation of the label is determined from the points P0 and P1. Capital letters and numbers are supported. Small characters are not supported. The following special characters are allowed: .(dot), -(minus), line separator is |
- **Via Fence** Creates a via fence between a set of points. Via diameters, via distance, catch pad width are defined in the Parameters. This object can define several groups and staggered vias can be set.

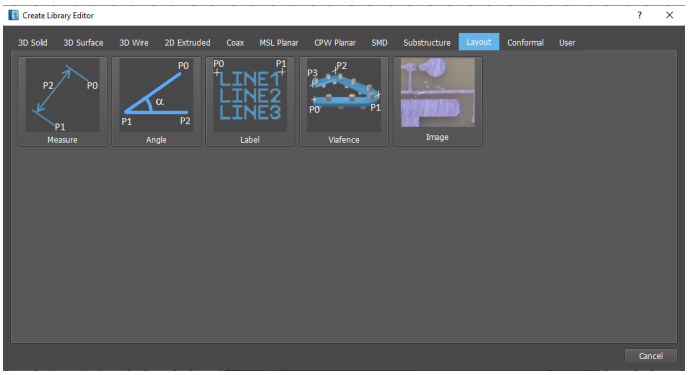


Figure 15.19.: View of Layout Library

- **Image**, 2D Design, only. Insert a bitmap image from a user supplied file into the background, e.g. to be used as a template for drawing.

Hint: Application Note

More information on Via Fence is available in Chapter 29.2.23

15.2.11. Conformal

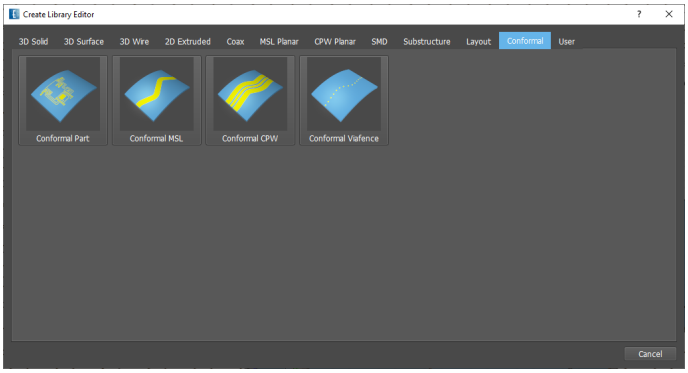


Figure 15.20.: View of Conformal Library

A collection of conformal objects for modeling of non-planar HF structures on arbitrary 3D surfaces. It is assumed that a bent surface object is already available. Then the bending is determined by clicking the button **Set Reference Object** in the library editor, clicking on the respective bent surface object and pressing the **Return** key or the button in the editor again.

- **Conformal Part** Map a planar layout (EMPIRE XPU™ legacy input file ending **.gym**) onto a bent surface. See section 15.2.9 for details
- **Conformal MSL** Create a microstrip line on a bent surface. Input points are start, rounded corners and end of the microstrip. With **Edit Settings** the geometry and resolution can be set. Mesh hints can be set to account for the specific bending properties.

- **Conformal CPW** Create a coplanar line on a bent surface. Input points are start, rounded corners and end of the CPW. With **Edit Settings** the geometry and resolution can be set. Mesh hints can be set to account for the specific bending properties. As special features the line can be buried and can be accompanied with a via fence for grounding which parameters can be set in the editor.
- **Conformal Viafence** Create a viafence on a bent surface. Input points are start, rounded corners and end of the fence. With **Edit Settings** the geometry and resolution can be set. As a special feature the vias can be equipped with pads.

Hint: Application Note

More information is available in Chapter 29.2.3

15.2.12. User

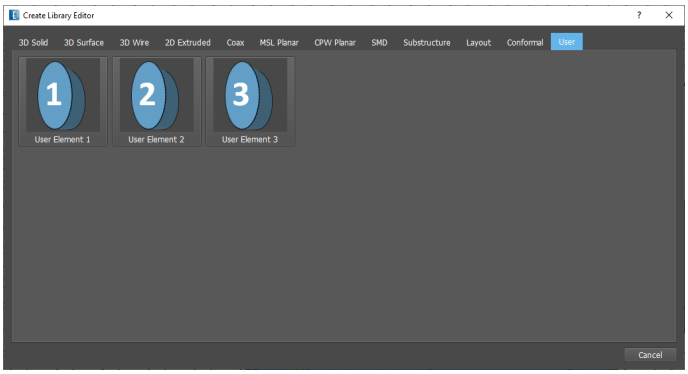


Figure 15.21.: User Defined Object Library

A collection of user defined objects for individual library extension.

With some programming skills the library can be extended by adding individual objects. It is best practice to adapt an existing similar library object code and to add the element into this tab.

An example is described in chapter 28.1.2

16. Object modification

This chapter lists a choice of operations which are often used in EMPIRE XPU™. A complete list of all available operations and their explanations can be found in chapter 33.

16.1. Move Operations

16.1.1. Move Object

Forward usage

1. Press the button **Move** 
2. Select objects to be shifted
3. Enter the shift arrow

Backward usage

1. Enter the shift arrow(s)
2. Select objects to be shifted
3. Press the button **Move** 

Mouse input (2D Design)

1. Select object(s) to be shifted
2. Move the mouse to one green handle
3. Press and hold the **right** mouse button
4. Drag the mouse to the desired position

16.1.2. Move Object to Group

If objects have to be moved from one group to another the following procedures can be applied:

1. Select object(s) to be moved to another group
2. Right click on group
3. Select  **Move selected objects to this group**

16.1.3. Parametric Move

For geometry variations it is possible to move objects parametrically. Here, the shift vector may be defined with parameters.

1. Select objects to be shifted parametrically
2. Press **Advanced**  **Parameters** and **Move Parameter** 
3. Enter shift arrow, e.g. **x=xvar y=yvar** and press **OK**
4. Enter **Min Max Value Step** for the parameters and press **OK**

Now the parameters are available in the **Variables** list and the range can be verified by using the sliders.

16.1.4. Mirror

Objects can be mirrored with the aid of an arrow which is parallel to one of the coordinate axes. Grid normal and arrow define the mirror plane.

1. Select the grid
2. Select objects to be mirrored
3. Click **Mirror** 
4. Enter mirror plane coordinates and press **OK**

- Instead of entering coordinates the arrow can be entered by mouse input with **Pick coordinates**.
- The 1D arrow represents the tangent of the mirror plane.
- The length of the arrow can be arbitrary but must not be zero.

16.1.5. Rotate

Objects can be rotated with the aid of a point on the current grid. Grid normal and point define the rotational axis.

1. Switch to the desired drawing plane
2. Select objects to be rotated
3. Click **Rotate** 
4. Enter point coordinates and press **OK**
5. Enter rotation angle and press **OK**

- Instead of entering coordinates, a point can be entered with **Shift** and left mouse click in the drawing area.
- The sequence of the steps can also be mixed.
- The default rotation angle can be set in the **Editor Options - Design - Rotation Angle**

Hint: Multiple Rotation

Multiple rotations can be specified with `angle@number`

16.1.6. Align

Objects can be centered to a point or arrow which can be helpful to align objects.

1. Switch to the desired drawing plane
2. Select objects to be centered
3. Click **Advanced** **Transform** and select **Align object to Point**
4. Enter point coordinates and press **OK**

- Instead of entering coordinates, a point can be entered with **Shift** and left mouse click in the drawing area.
- The sequence of the steps can also be mixed.
- It is also possible to center objects to the midpoint of an arrow by using **Align object to arrow**
- The alignment can also be restricted in a single direction by pressing the associated icon.

16.1.7. Transform via LCS

If a Local Coordinate System (LCS) has been defined objects can be moved and rotated from the LCS to the World Coordinate System (WCS) applying a mapping between u,v,w and x,y,z origins and vice versa.

1. Select objects to be transformed
2. Click **Advanced** **Transform** and select **Transform from LCS to WCS** or **Transform from WCS to LCS**

16.2. Copy Operations

16.2.1. Copy Object

Hint: Copy Objects

Copied objects will always be generated in groups of the original objects.

Forward usage

1. Press the button **Copy**
2. Select objects to be copied
3. Enter the shift arrow

Backward usage

1. Enter the shift arrow(s)
2. Select objects to be copied
3. Press the button **Copy**

Mouse input (2D Design)

1. Select object(s) to be copied
2. Move the mouse to one green handle

3. Press and hold the **middle** mouse button
4. Drag the mouse to the desired position

16.2.2. Copy to Current Group

1. Select object(s) to be moved to another group
2. Right click on group
3. Select **Move selected objects to this group**

16.2.3. Array Copy

For the generation of an array this operation is useful.

1. Select object to be copied multiple times
2. Click the button **Multiple Copy**
3. Select between rectangular or linear array
4. Enter distances and number of array elements **OK**
5. Optionally enter a mapping function using x, y or z as variable.

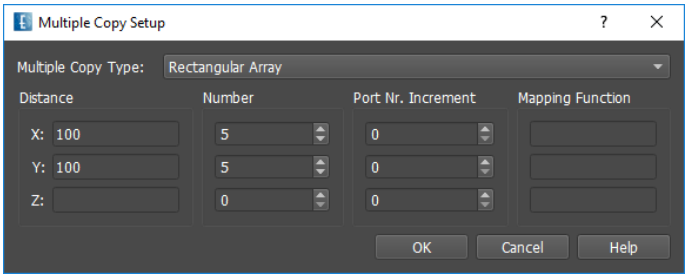


Figure 16.1.: Array copy wizard

Mapping Example

A mapping function can be used, e.g. to create a circular array from a selected object. Figure 16.2 shows an example which creates a circular array around a center of the the selected object with radius defined by variable a, step angle defined by distance x=30° and 12 copies (Number=13). Note that the selected object is maintained as in all copy operations.

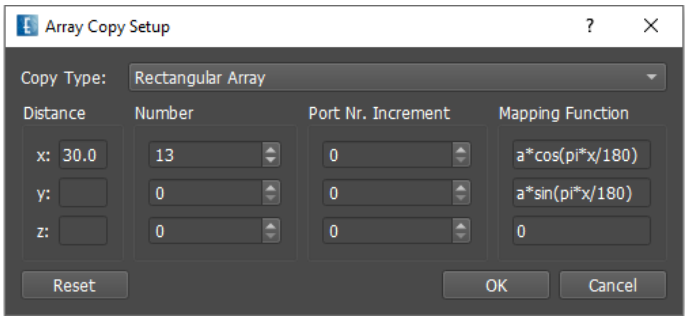


Figure 16.2.: Mapping example for array copy

16.2.4. Copy & Mirror

Similar to **Mirror** but the original object will be maintained.

1. Switch to the desired drawing plane
2. Select objects to be mirrored
3. Press **Copy & Mirror**
4. Enter mirror plane coordinates and press **OK**

16.2.5. Copy & Rotate

Similar to **Rotate** but the original objects will be maintained.

1. Switch to the desired drawing plane
2. Select objects to be rotated
3. Press **Copy & Rotate**
4. Enter point coordinates and press **OK**
5. Enter rotation angle and press **OK**

Hint: Multiple Rotation

Multiple rotations can be specified with `angle@number`

16.3. Stretch Operations

16.3.1. Stretch Object

Forward usage

1. Press the button **Stretch**
2. Select objects to be stretched and press **OK**
3. Activate handles (green dots) of corners or edges to be shifted

4. Enter the shift arrow and press **OK**

Backward usage

1. Select objects to be stretched
2. Activate handles (green dots) of corners or edges to be shifted
3. Press the button **Stretch** 
4. Enter the shift arrow and press **OK**

Mouse input

1. Select objects to be stretched
2. Activate handles (green dots) of corners or edges to be stretched
3. Move the mouse to one green handle
4. Press and hold the **left** mouse button
5. Drag the mouse to the desired position

16.3.2. Parametric Stretch

For geometry variations it is possible to stretch objects parametrically. Here, the stretch vector may be defined with parameters.

1. Select objects to be stretched parametrically
2. Press **Advanced**  **Parameters** and select **Stretch Parameter** 
3. Enter stretch arrow, e.g. **x=xvar y=yvar** and press **OK**
4. Enter **Min Max Value Step** for the parameters and press **OK**

Now the parameters are available in the **Variables** list and the range can be verified by using the sliders.

16.3.3. Scale

Objects can be scaled to their center or to the origin (circles on grid bar).

1. Select objects to be scaled
   - Press **Advanced**  **Transform** and select **Scale to Center** 
   - Press **Advanced**  **Transform** and select **Scale to Origin** 
   - Press **Advanced**  **Transform** and select **2D Scale to Origin** 
2. Enter scale factor and press **OK**

Hint: Object scaling

If the scaling should not be performed in the perpendicular direction, the **2D Scale to origin** can be used.

16.3.4. Oversize

Objects can be enlarged by a certain amount using the function oversize.

1. Select objects to be enlarged
2. Press **Advanced** **Transform** and select **Oversize**
3. Enter a number in units (may be negative to undersize)

16.3.5. Rasterize Polygons

Imported layout data exhibit sometimes odd coordinate numbers. Tthe function **Rasterize** can help to align edges and vertices to a desired grid.

1. Select polygon to be aligned
2. Press **Advanced** **Polygon** and select **Rasterize**

The properties of the rastering grid can be adjusted in **Editor Options - Components - Polygon Rasterize**

16.4. Change operations

16.4.1. Set radius

Converts a polygon with sharp edges into a polygon with rounded edges.

1. Select Polygon
2. Activate handles on corners which should be rounded
3. Press **Advanced** **Polygon** and select **Set radius**
4. Enter radius in units

16.4.2. Convert Boxes

In some cases it can be useful to convert box objects into another object types. Select Box, click **Advanced** **Edit - Convert** and select

- **Group Boxes** combines several boxes into one multibox.
- **Convert to Polygon** to be used if, e.g. corners should be rounded
- **Convert to Solid** Surface representation of box
- **Convert to Lumped Port** Create a (perpendicuar) lumped port with the size of the box and direction determined by a plane:
  - **2D Design** Perpendicular to the current view
  - **3D Design** Perpendicular to the current grid
- **Split Object to Faces** Explode box into 6 singe faces

16.4.3. Convert Polygons

Select Polygon, click **Advanced**

- **Linpolys & Bond Wires - Create Bondwire** (for 2 point polygon) yields a linear polygon with a bondwire shape
- **Linpolys & Bond Wires - Create Line poly** yields an open-ended linear polygon (general wire)
- **Convert - Convert to Bounding Box** Convert to bounding box
- **Convert - Convert to Solid** Boundary representation of polygon

16.4.4. Convert Solid

Imported solids can exhibit properties which are non-advantageous for a 3D modeler, such as zero thickness, no orientation, etc. Some operations can help to work with those objects nevertheless. Select Solid, click **Advanced**

- **Solid - Heal Solid** Tries to heal a solid which has defects.
- **Solid - Simplify Solid** Tries to optimize solid by reducing the number of triangles
- **Solid - Extrude Flat Solid** Solids with zero thickness will be extruded in normal direction to obtain a closed surface with adjustable thickness
- **Edit - Convert - Split Solid** Split solid into parts
- **Edit - Convert - Convert to Polygon** The selected solid object is transformed to a closed polygon where the following input will be requested.
  - **Cutting Plane** Defines the slicing plane
  - **Tolerance** Accuracy of the cut
  - **New height** Extension of the polygon
  - **Keep object's groups** Delete unused groups
  - **Merge resulting polygons** Join resulting polygons
  - **Simplify resulting polygons** Remove off-corner points
- **Convert to Bounding Box** The solid object is transformed to its bounding box

16.5. Boolean operations

In order to create more complex shapes, Boolean operations between all objects are allowed. If necessary, objects will be converted into solids.

1. Select two or more objects to apply Boolean operations
   - **Union** Merge objects
   - **Subtract (first-rest)** Subtract the all but first objects from the first
   - **Intersect** Create common volume from 2 objects
   - **Add** Merge object without simplification
   - **Subtract (first-rest, keep rest)** As above, but objects are maintained

- **Multiple Subtract (largest-rest)** Useful to create multiple holes.
- **Multiple Subtract (rest-last)** Useful to cut several objects with the same object.

Hint: Boolean history

If solids contain variables they will be lost after a Boolean operation by default. In order to keep the variables a Boolean history can be enabled in the **Editor Options** in **Preferences**

Hint: Defective solids

In EMPIRE XPU™ Solids are built from small triangles which enclose an arbitrary shaped body and describe a closed surface. If the surface is not closed (which can happen during import of external data) the solid is marked as defective. This is usually no problem for the simulation but Boolean operations can not be executed. The heal function can help in this case.

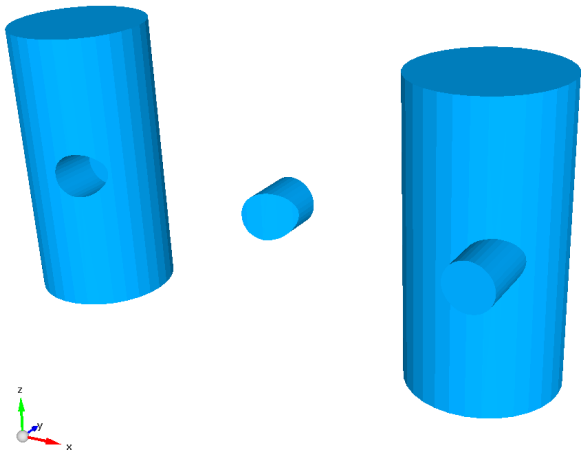


Figure 16.3.: Examples of Boolean operations Subtract, Intersect and Merge

Remarks

- If the input objects are polygons with the same height, the output will be a polygon, too.
- If the input objects are polygon with different heights, the user will be asked for the output type:
  - **Polygon** The output is sliced into different polygons (boundary representation BREP is kept)
  - **Solids/Surfaces** The output is a triangulated surface (FACET, no BREP)
- Preferred handling can be set in **Editor Options - Preferences - Boolean Operations**

17. Import and Export

This chapter gives an overview on how to import data into EMPIRE XPU™ and export from EMPIRE XPU™. After import, operations like shaping, drilling, merging, are still possible as described in chapter 16.

17.1. Import

By clicking the import button  a source file containing the data can be selected. The following described formats are supported and can be also imported with the **File - Import** menu. If necessary some import options can be set in advance in the  **Editor Options - Import** menu.

17.1.1. Advanced

These import options support files with specialized formats.

Empire

- **Empire Meta File** Import a classic meta file (ending .gmf), e.g. created by a script
- **Empire KMeta File** Import a meta file (ending .kmf), e.g. created by a script
- **2D Points from Python Script** Import of a point list into the current 2D Design plane, defined in a script

Data

- **Import R&S CUT data** Reads farfield data of R&S .cut files and stores in Empire readable format in subfolder `v_RS-Data`. A folder where the .cut file is located has to be selected during import.

2D Layout

- **Asitek** Import of a layer stack (including name, height) with a format convention defined by company Asitek Oy
- **DSN** PCB layout data exported from routing software
- **EAGLE-Excellon** Excellon-like format for PCB drilling and routing machines
- **Excellon** Format for PCB drilling and routing machines
- **HP-GL** Printer control language created by Hewlett-Packard (HP)

Hint

The **Excellon** import assumes the file extension **.drl** or **.ncd**. If drill files are ending differently they should be renamed before import.

3D Solids

- **Import Collada** (ending .dae)
  Interchange format for interactive 3D applications, based on 3D polygonal meshes.
- **Run Tool for Large Data Import**
  An import tool is available which is dedicated to large CAD models, e.g. vehicles. The built-in algorithms reduce the complexity of the CAD data and generate equivalent models without impairing the electrical behavior for the targeted EM simulation. The resulting dataset is strongly reduced in size to be handled easily by 3D EM simulation tools. More info is available in Chapter 4.1.4 of **Getting Started EM-Twin.pdf**

17.1.2. Empire

These import options support EMPIRE XPU™ input files.

- **from Clipboard** Import selections from another project which are copied to Clipboard by **Ctrl c**
- **from File** Import project input file, ending .gym, .emx and .emt
- **2D Points from File** ASCII file with a list of coordinate points (2 columns)

The latter can be used to create special shapes by importing a set of points from a simple list which has been created, e.g. by another tool. After import the points will be displayed in EMPIRE XPU™ and can be further processed, e.g. to create a polygon with the cross section defined by the points. The points and the polygon cross section will be generated in the current drawing plane. Therefore, the desired plane has to be selected first.

The file can have any name, e.g. **list.txt** and the entries have to be plain ASCII. One example of a parabolic shape could be:

0	0
100	10
200	40
300	90
400	160
600	360
800	640
1000	1000
1000	-100
0	-100

The procedure to generate this parabolic cylinder is:

1. Select Menu **File - Import - Empire - 2D Points from File**
2. Browse directory and select point list file
3. Press button **Create Poly** 

17.1.3. EDA

Files for Electronic Design Automation contain not only information about geometry but can also contain electric properties or SMD components.

- **Import ODB++ (File)** - compressed folder ending **.tgz**
- **Import ODB++ (Directory)** - uncompressed folder

ODB++ is a proprietary CAD-to-CAM data exchange format which purpose is to exchange printed cicuit board design information. In contrast to **2D Layout Data** the **ODB++** contain height information, electric parameters and SMD components.

During Import, a wizard displays all available layers with height and electric property information. Support layers can be deselected before import.

Hint

The imported geometry is created and merged as polygons. Especially holes should be checked for integrity and the internal line angle should be adjusted if necessary, see 15.1.3

17.1.4. 2D Layout Data

Imported layout objects are always represented as polygons. Depending on the current view they are imported in xy- (Top view) yz- (Front view) or xz- (Right view) plane. Layout data is available in files which have a certain standard. EMPIRE XPU™ supports the following file types:

- **GDSII** - ending **.gds**
- **Gerber** - ending **.ger**, **.gbr** (supported version Gerber 247x)
- **2D DXF** - ending **.dxf** (supported version DXF 12)

Hint: Drill files

NC drill files (e.g. ending .TXn) can be imported in **Import - Advanced - 2D Layout - Import Excellon/NC Drill**

Before import, the respective settings in the **Editor options** should be checked and adjusted. The most important settings are:

- **Import Scale Factor** - To be adjusted if layout unit and EMPIRE XPU™ drawing unit are different
- **Arc resolution** - If a layout contains a huge amount of curved objects the import time can be reduced by enlarging the default resolution.
- **Circle recognition** - Automatically convert multiple point circles into single point circles to speed up drawing
- **DXF connection tolerance** - A value > 0 can be set to join single lines into polygons.

The import procedure is executed with the following commands:

1. Select Menu **File - Import - Layout** and file format or press the **Import** button 
2. Browse directory and select layout file
3. Optionally, select or deselect layer or additional files to be imported
4. Optionally, adjust **Object Unit** to fit layout to the current unit and press OK
5. Optionally, select layer, rename, set height and property in the import wizard
6. Press **Import** and **Close** to finish the layout import
7. If not done in the wizard, browse through the groups and assign a respective height and electric property

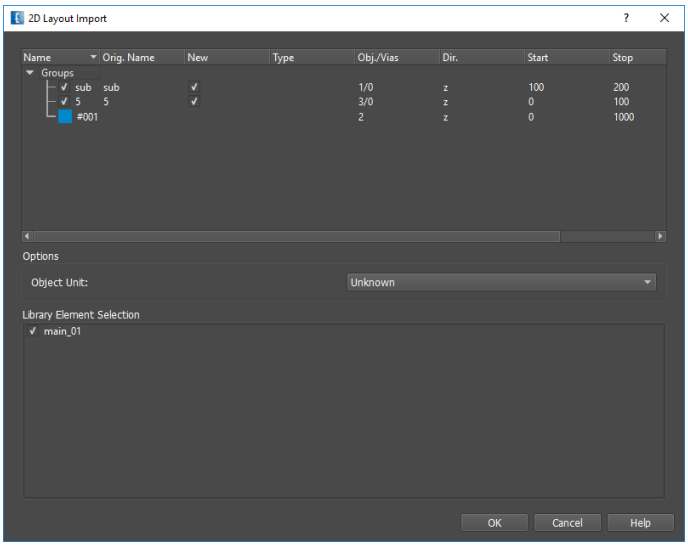


Figure 17.1.: Import wizard for DXF

Hint: Layout import

Layout files do not contain any height information. This has to be set by the user. If a certain layer stack has been set once it can be re-used if the group names are identical for another layout file.

Hint: Import Map

A layer stack can be exported and re-used by clicking **Import Map**. Imported layers will be listed in the **Groups** tree. They can be mapped to existing layers using drag and drop.

Hint

The imported geometry is created and merged as polygons. Especially holes should be checked for integrity and the internal line angle should be adjusted if necessary, see 15.1.3

Hint: Application Note

More information is available in Chapter 29.2.12

17.1.5. 3D Solid Data

EMPIRE XPU™ provides import for all common 3D-CAD-formats, enabling users to directly load geometries created with other CAD- or CAM-tools into the EMPIRE XPU™ workspace. Portions of this software are owned by Spatial Corp. © 1986 – 2024. All Rights Reserved. The following formats are always supported:

- **NASTRAN** (.nas) popular automotive format
- **OFF** (.off) Geometry definition file format containing the description of the composing polygons of 3D objects
- **STL** (.stl) Stereo-Lithography, native solid format in EMPIRE XPU™

The following formats require a corresponding license:

- **ACIS** (.sat, .sab, .asat, .asab, .sm3) native format of ACIS-modeling-kernel by Spatial (affiliated to Dassault Systèmes)
- **CATIA V4 / V5** (.model, .exp, .session) format by Dassault Systèmes, commonly used in automotive and aerospace engineering
- **DXF(3D)** (.dxf) 3D format by AutoCAD
- **IGES** (.igs, .iges) intermediary format
- **Inventor** (.ipt, .iam) format by Autodesk
- **JT** (.jt) Jupiter Tesselation format by Siemens PLM Software
- **Parasolid** (.x_t, .xmt_txt, .x_b, .xmt_bin) native format of Parasolid-modeling-kernel by Siemens PLM Software
- **Pro/E / Creo** (.prt, .prt., .asm, .asm.) PTC Creo format, formerly Pro/Engineer, by Parametric Technology Corporation
- **Siemens NX** (.prt) format by Siemens, commonly used in automotive engineering
- **Solid Edge** (.par, .asm, .psm) by Siemens PLM Software
- **SolidWorks** (.sldprt, .sldasm) by Dassault Systèmes
- **STEP** (.stp, .step) commonly used intermediary format, intended as successor for IGES and VDA-FS

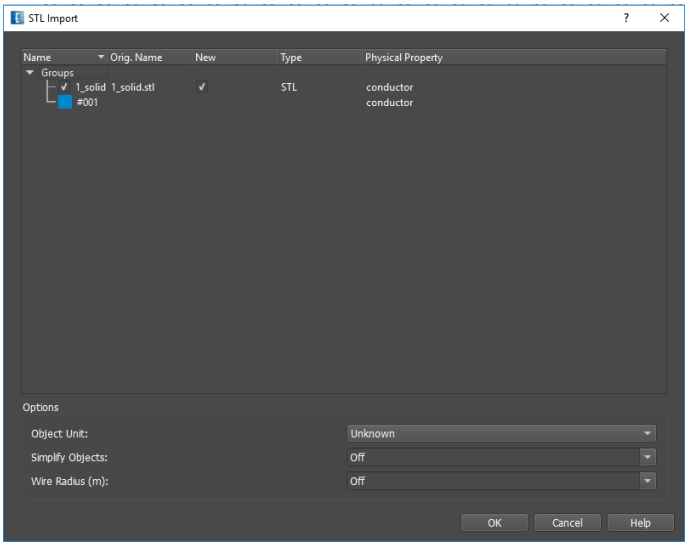


Figure 17.2.: Import wizard for STL

Before importing *.stl- or *.off-files, the respective settings in the **Editor options** should be checked and adjusted. The most important settings are:

- **Scaling** - To be adjusted if drawing units are different
- **Import Shift** - To be set if origins are different
- **Separate STL objects** - In some cases several solids are grouped in one file. In case of ASCII STL, the separation is done automatically. In case of binary STL they can be separated if switched **On**

- **STL Resolution** - Reduce complexity of solids to speed up import and drawing time.

The import procedure for *.stl- or *.off-files is executed with the following commands.

1. Select Menu **File - Import - 3D Solids - Import STL object** or press the **Import button**
2. Browse directory and select file with ending **.stl**
3. Select or deselect additional files to be imported
4. Press **Import** and **Close** to finish the import
5. Browse through the groups and assign a respective electric property

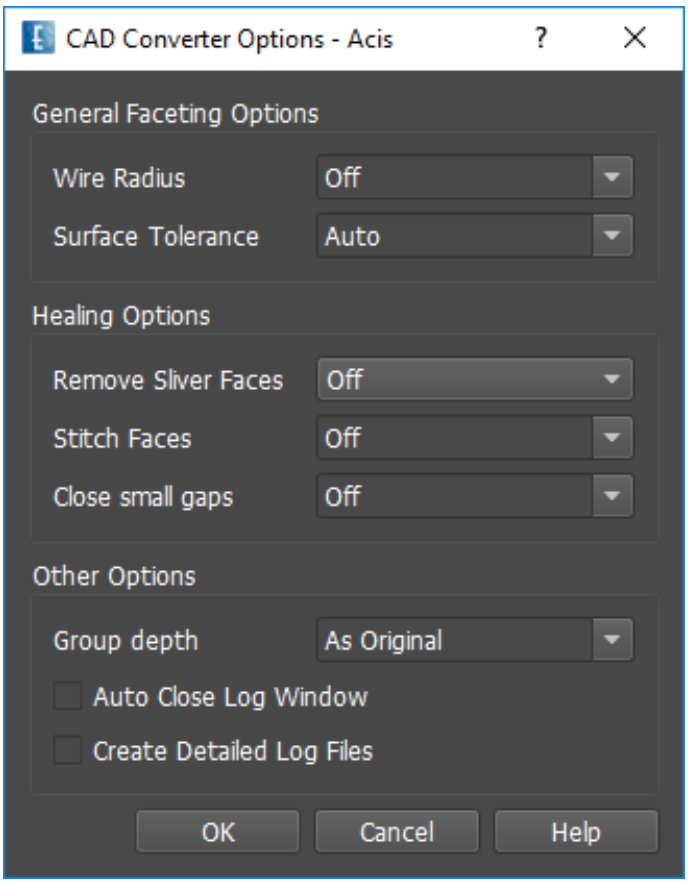


Figure 17.3.: CAD Converter Options for ACIS, CATIA, NX, etc.

While importing other 3D-CAD-files, e.g. *.sat, a dialog named “CAD Converter Option” appears. Its settings should be checked and adjusted:

- **Wire Radius** - Import B-REP-wire-bodies with radius (m)
- **Surface Tolerance** - Acceptable surface deviation between original (B-REP) and imported (discretised) data, given as a factor of target wavelength.
- **Remove Sliver Faces** - Sliver-like faces with high ratios of ”longest-edge/shortest-edge” will be removed.
- **Stitch Faces** - Faces will be stiched together to form complete bodies. If the maximum stitching tolerance is given as an absolute value (in drawing unit), it should be smaller than the minimum feature size and bigger than the maximum gap expected to be stitched in the model.

- **Close small gaps** - If enabled, small gaps will be closed. If the desired gap tightness is given as an absolute value (in drawing unit), it should be set to the desired manufacturability tolerance.

The import procedure is executed with the following commands.

1. Select Menu **File - Import - 3D Solids - Import FORMAT** or press the **Import button** 
2. Browse directory and select file
3. Press **OK** to start the import
4. Adjust settings in "CAD Converter Option" dialog (see above)
5. Press **OK** to finish the import
6. Browse through the groups and assign a respective electric property

Hint: Application Note

More information is available in Chapter 29.2.2

17.1.6. CST-Projects

It is possible to migrate projects from CST - Microwave Studio into EMPIRE XPU™. Simulation settings are automatically transferred.

The import of CST-projects into EMPIRE XPU™ is executed with the following commands.

1. From CST-software, export geometry as *.stl and save it in the folder of the project file using the same file prefix.
2. Select Menu **File - Import - Projects - Import CST**
3. Browse directory and select the project file ending **.cst**
4. Press **Close** to finish the import

An info window is displayed which shows the import actions. Simulation settings, mesh lines, material and objects are imported in the current EMPIRE XPU™ file.

The following files are analyzed and used for import:

- `<PROJECTPATH>/<PROJECTNAME>.stl`
- `<PROJECTPATH>/<PROJECTNAME>/Result/Model.msh`
- `<PROJECTPATH>/<PROJECTNAME>/Model/3D/Model.par`
- `<PROJECTPATH>/<PROJECTNAME>/Model/3D/Model.mod`

Port definition and setup have to be added manually. It is recommended to browse the group tree and verify the objects and properties.

Preserved Features

- Geometry
  - Grouping of components
  - Colors of objects
- General settings
  - Drawing unit
  - Frequencies
  - Boundary conditions
- Materials
  - Background material
  - Parameter based materials
- Lumped Elements, e.g. SMD Resistors
- Excitation
  - ”Discrete port” and ”Discrete Face Port”
  - Placeholders for incompatible ports
- Mesh
- Far-field monitors

Tips and Tricks

- Best import results can be expected if
  - No local coordinate systems are used
  - Only ”DiscretePort” or ”DiscreteFacePort” are used
  - Project was created without copy & paste
- Import is based on project history. Re-edit detais and save may help in case of import problems
- Enable world coordinate system before export STL solids
- View import log file for possible errors and warnings

Hint: Placeholders

In general ports are imported as boxes which act as placeholders. For lumped ports a function is available to easily create lumped ports from the placeholders, see Chapter16.4.2.

Hint: Application Note

More information is available in Chapter 29.2.5

17.1.7. HFSS-Projects

It is possible to migrate projects from AEDT into EMPIRE XPU™. Simulation settings are automatically transferred. An import license for **ACIS** files is needed.

The import of HFSS-projects into EMPIRE XPU™ is executed with the following commands.

1. From HFSS-software, export **ACIS** geometries as *.sat, *.sab or *.sm3 and save them in the folder of the project file (.hfss or .aedt)
2. Select Menu **File - Import - Projects - Import HFSS**
3. Browse directory and select the project file
4. Press **Close** to finish the import

An info window is displayed which shows the import actions. Simulation settings, materials and objects are imported into the current EMPIRE XPU™ file. The user may be prompted for certain decisions during import, e.g. which HFSS-Model to import (if there are more than one in the HFSS-file). Port definition and setup, as well as mesh have to be added manually. It is recommended to browse the group tree and verify the objects and properties.

Import Highlights

- Support of projects with multiple:
  - Models
  - Solver setups
  - Sweep configurations
- Preserves the following features during import:
  - Geometry including object color
  - General settings, e.g. drawing unit, frequency range, target frequency, boundary conditions
  - Materials
    - Parameter based materials
    - Anisotropic materials
    - Sheet materials
  - Lumped ports with integration line

Tips and Tricks

- Best import results can be expected if only ports of type “Lumped Port” are being used
- In case of incompatible ports, placeholder will be created to aid manual reconstruction
- Try different settings for “Surface Tolerance” during geometry import
- View import log file for possible errors and warnings

Hint: Placeholders

In general ports are imported as boxes which act as placeholders. For lumped ports a function is available to easily create lumped ports from the placeholders, see Chapter16.4.2.

Hint: Application Note

More information is available in Chapter 29.2.10

17.1.8. Data - Import Touchstone

Import of S-parameter data files in Touchstone Format (ending .snp) to be displayed in the **2D Results** Tab.

During import a subfolder is created in the current project folder and single S-parameters are extracted which are listed in the **Plot - Legend**

17.2. Export

The geometry can be exported in several formats.

17.2.1. Advanced

Export of solids or layout in special formats.

2D Layout

- **Excellon** format used to drive CNC drilling and routing machines in printed circuit board (PCB) manufacturing
- **HP-GL** printer control language created by Hewlett-Packard (HP)
- **MLS** format (7032) used for IMST’s laser drilling machine
- **UHT** format used for IMST’s LTCC punching machine
- **Layer Stackup (CSV)** Export layer information in a comma separated file
- **SMD Info (CSV)** Export SMD information in a comma separated file

3D Solids

- **Collada** interchange file format for interactive 3D applications used, e.g. for EMPIRE XPU™ VMI module
- **IDTF** Intermediate Data Text Format. It is a text, intermediate file, used as a step before the creation of the U3D files
- **U3D** export for creating 3D-PDFs from EMPIRE XPU™ models
- **U3D + TeX** as above, additional Tex environment

17.2.2. 2D Layout

Exported layout objects are always represented as polygons. Depending on the current view they are exported from xy- (Top view) yz- (Front view) or xz- (Right view) plane. EMPIRE XPU™ supports the following file types:

- **DXF 12** - ending **.dxf** (supported version DXF 12)
- **GDSII** - ending **.gds**
- **Gerber** - ending **.gbr** (supported version Gerber 247x)
- **PDF** - ending **.pdf** used to create layout images as vector graphic
- **PCAD** - ending **.pcb**

The export procedure is executed with the following commands.

1. Select Menu **File - Export - Layout** and file format
2. Browse directory and enter a file name

Hint: GDS Export

The group names have to be numbers in order to export GDS data

Hint: Gerber Export

For each group a separate file will be generated

Hint: PDF Export

The line width is defined in Empire units. If the line width appears to thick the **PDF Scale Factor** should be enlarged in the **Editor Options - Export Advacned**

Hint: Application Note

More information is available in Chapter 29.2.11

17.2.3. 3D Solids

The following export formats are supported as **FACET** representation and included in EMPIRE XPU™ standard packages:

- **NASTRAN** (.nas) popular automotive format
- **OFF (ASCII)** Object File format
- **STL (BIN)** Stereo Lithography in binary format
- **STL (ASCII)** Stereo Lithography in plain text format

STL File Format (.stl) files are used to represent the geometry of a model by specifying triangles of the model's surface. Object File Format (.off) files are used to represent the geometry of a model by specifying the polygons of the model's surface. The polygons can have any number of vertices. Also the edge relations of neighbour facets are contained in the OFF, while this information is missing in the STL format.

The following list of CAD standards are supported as **BREP** and **FACET**[1] representation, but require valid licenses:

- **ACIS** (ASCII .sat or BIN .sab) compatible format to geometric modelling kernel developed by Spatial
- **CATIA** (v4 .model or v5 .CATPart) format by Dassault Systèmes, commonly used in automotive and aerospace engineering
- **IGES** intermediary format
- **STEP** commonly used intermediary format, intended as successor for IGES and VDA-FS

The export procedure is executed with the following commands.

1. Select solids which have to be exported
2. Select Menu **File - Export - 3D - (Format)**

Hint

The native format of EMPIRE XPU™ solids is STL which is a triangular surface description of the enclosed volume (**FACET**). The exported formats here keep this description. An export of boundary representation (**BREP**) is available in the **Editor Options - Export** menu

The **BREP** export engine can be selected in the **Editor Options - Export - 3D Solids**:

- **Open CASCADE** Geometry exported natively as **BREP** where possible. Uses primitive planar approach (see below) as fallback. Requires target format license and **STEP** license
- **Primitive Planar** Geometry exported effectively as facets, i.e. one planar **BREP** face per **FACET**. Requires target format license
- **Polyhedra** Geometry exported as **FACETs** and then converted to **BREP** where possible using the CGM Polyhedra module. Uses primitive planar approach as fallback. Requires target format license and **PHD** license

[1] Each facets described as one planar BREP face

17.2.4. Empire Part

Creates the native EMPIRE XPU™ input file format.

- **to Clipboard** Export Selections to Clipboard (Same as **Ctrl c**)
- **to File** Export Selections to a project file, ending .gym

17.2.5. Read Protected File

This file is used to exchange simulation files without revealing geometry, material properties, ports and meshing. Field monitors, simulation parameters or boundary conditions are not exported. The file can be imported with the aid of a substructure library element **Protected Part** see 15.2.9.

Export options:

- **Read Protected Simulation File** Creates a file ending **.aprot**. Before exporting it is possible to apply an encryption key for increased security.
- **Encryption Key File** Generates an encryption key file ending **.akey**

If the encryption has been applied in export both files are needed to be able to import the substructure library element and to perform the simulation.

17.2.6. Export Image

Create an image in portable network graphics (PNG) of the current display.

18. Object properties

To all objects certain properties have to be assigned to. They can be either physical, i.e. dielectric, or functional properties, i.e. mesh hint or animation. The object properties are derived from the group properties. It is possible to define more than one property to a group if they are not contradicting. For example, a stripline can be defined to be a conductor and obtain a mesh hint but it may not be defined as conductor and dielectric.

Some properties may also be defined with variables, e.g. for variation and optimization purposes.

Hint: Parametric properties

By entering a variable name, e.g. `anyname` instead of a number the value can be controlled in the variable list in the left tree. If the variable is new, its type, range, current value and step size can be set.

Hint: Set property

In order to set a property of a group, double click on the property item or use the **Edit Property** button of the group context menu.

18.1. Priorities

In the case that objects overlap which have different physical properties a priority number (integer value between 10 and 250) can be defined for each physical property.

Hint: Default Priority

By default conductors have priority number 200 and dielectrics have priority number 100.

As a consequence, if conductors are created inside a dielectric volume, they are recognized in the simulation without the need of cutting out the dielectric.

If a conducting block should be cut out by a dielectric volume it is sufficient to assign the dielectric a higher priority number than the conducting block.

This concept can be applied for multiple overlapping objects. As an example, a coaxial line could be made of

- Large cylinder, conductor, priority 50
- Medium cylinder, dielectric, priority 100
- Small cylinder, conductor, priority 150

Hint: Surface Priority

The priority concept is also applied to surfaces, after mapping to the mesh.

As a result, flat metal can be disregarded from a simulation if the priority number is smaller than the one of the substrate carrier.

18.2. Material Property

Objects inherit the property of their group. Mesh cells which are covered by those objects are filled with this material.

18.2.1. Dielectric

It is recommended to use this property for the definition of general dielectrics with constant values. The loss model which is set in the simulation setup in Section 23.2.3 applies.

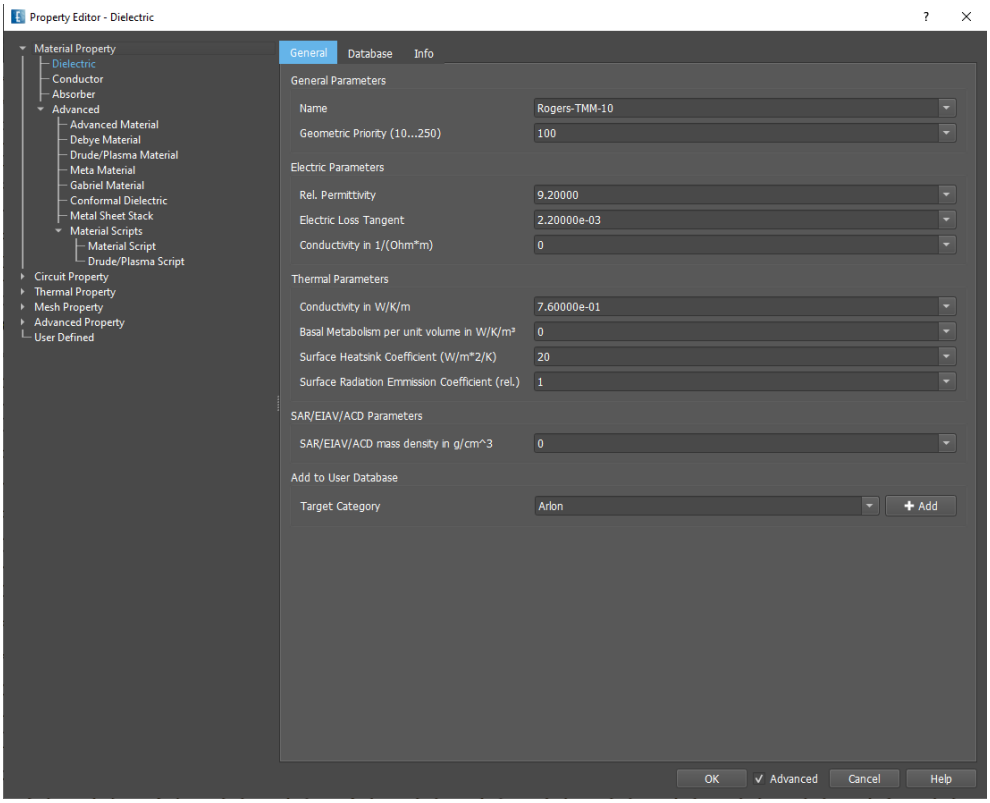


Figure 18.1.: Dielectric property setup

Tabs

- **General** - Here, the material properties can be entered manually.
- **Database** - A list of predefined materials can be selected for usage (extendable).
- **Info** - The frequency dependence is realized by using high order Debye models and previewed in this window.

General

- General Parameters
  - **Name** - A unique name of an item in the data- and userbase or userdefined
  - **Geometric Priority** $10 < Integer < 250$, used for intersecting objects with different materials

- Electric Parameters
  - **Rel. Permittivity** - $\varepsilon_r \geq 1$
  - **Electric Loss Tangent** - Dissipation Factor $\tan\delta \geq 0$
  - **Conductivity** - $\sigma \geq 0$ in $\frac{1}{\Omega m}$
- Thermal Parameters
  - **Conductivity** Thermal conductivity $k \geq 0$ in $\frac{W}{Km}$
  - **Basal Metabolism per unit volume** in $\frac{W}{Km^3}$ used for body model heat sink
  - **Surface Heatsink Coefficient** Convection transfer coefficient $C_{HC} \geq 0$ in $\frac{W}{Km^2}$
  - **Surface Radiation Emission Coefficient** Emissivity $C_{Emiss} \geq 0$
- SAR/EIAV/ACD Parameters
  - **SAR/EIAV/ACD mass density** can be specified for averaging purposes $\rho \geq 0$ in $\frac{g}{cm^3}$
- Add to User Database
  - **Target Category** - A unique name of an entry in the database list (editable to create new category)
  - **Add** the settings will be stored in a file called `.ganymed_wizmat.txt` in the local user directory and can be reused in other projects.

Database

To use materials from the database, choose the entry by pressing ▶ and select one of the materials. Switch to **General** to verify the selection

Info

Switch to **Info** to see the frequency dependence of the selected material. It shows the graphs of the different dielectric models which can be set in Section 23.2.3.

18.2.2. Conductor

It is recommended to use this property for the definition of good conductors[1]. The loss model which is set in the simulation setup in Section 23.2.3 applies.

Hint: Conductors

By default, **Conductors** are modelled as flat and opaque sheet objects. To prevent this conductors can be assigned to be **bulk** objects in **Automesh Modelling** and in Section 23.2.3. Alternatively, lossy conductors can be defined as **Dielectrics** with certain conductivity, e.g. to model sea water.

Tabs

- **General** - Here, the material properties can be entered manually.
- **Database** - A list of predefined materials which can be selected (extendable).
- **Info** - The frequency dependence of thin sheet models models can be previewed in this window.

General

- General Parameters

[1] e.g. skin depth less than metal thickness

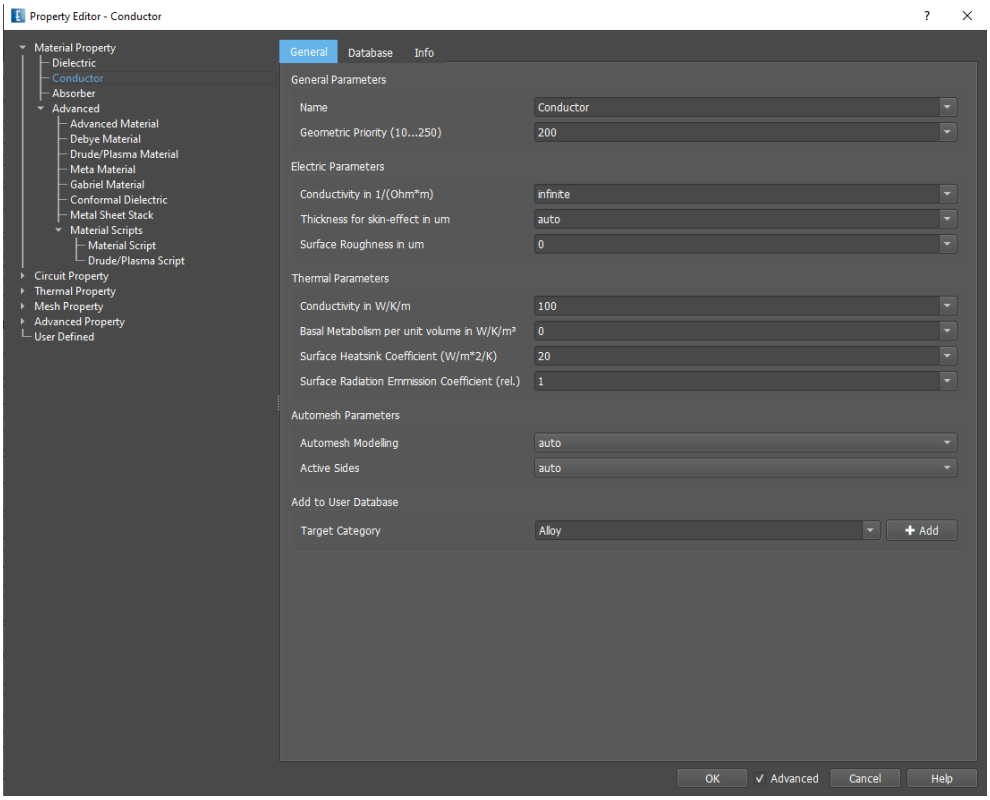


Figure 18.2.: Conductor property setup

  - **Name** - A unique name of an item in the data- and userbase or userdefined
  - **Geometric Priority** - $10 < Integer < 250$, used for intersecting objects
- Electric Parameters
  - **Conductivity** - $\sigma \geq 0$ in $\frac{1}{\Omega m}$
  - **Thickness for skin effect** - $0 < t < 10a$, in μm , can be determined automatically. [2]
  - **Surface roughness** - $0 \leq \rho < thickness$, average roughness in μm using the Hammerstad & Jensen model: $R^r_\square = R_\square \left(1 + \frac{2}{\pi} arctan(1.4\frac{\rho^2}{a^2})\right)$ where a is the skin depth and $R_\square$ is the square resistance at the target frequency.
- Thermal Parameters
  - **Conductivity** Thermal conductivity $k \geq 0$ in $\frac{W}{Km}$
  - **Basal Metabolism per unit volume** in $\frac{W}{Km^3}$ used for body model heat sink
  - **Surface Heatsink Coefficient** Convection transfer coefficient $C_{HC} \geq 0$ in $\frac{W}{Km^2}$
  - **Surface Radiation Emission Coefficient** Emissivity $C_{Emiss} \geq 0$
- Automesh Parameters
  - **Automesh Modelling** - auto-detected or set as **flat** or **bulk** material
  - **Active Sides** - If set to **groundplane** only one side is used for loss calculation.
- Add to User Database
  - **Target Category** - A unique name of an entry in the database list (editable to create new category)
  - **Add** the settings will be stored in a file called `.ganymed_wizmat.txt` in the local user directory and can be reused in other projects.

Database

To use conductors from the database, choose the entry by pressing ▶ and select one of the conductors. Switch to **General** to verify the selection

Info

Switch to **Info** to see the frequency dependence of the selected conductor. It shows the graphs of the different conductor models which can be set in Section 23.2.3.

18.2.3. Absorber

An absorber is a composed material with following model parameters in order to absorb a wave (a=Absorption in dB/mm)

$$\varepsilon_r = \varepsilon_2 \qquad \mu_r = \frac{\varepsilon_2}{\varepsilon_1} \qquad \sigma_e = \frac{Z_0}{\sqrt{\varepsilon_2}} a \frac{1000}{8.68} \qquad \sigma_m = \frac{\sqrt{\varepsilon_2}}{Z_0} a \frac{1000}{8.68}$$

- General Parameters
  - **Name** - A unique name of an item in the data- and userbase or userdefined
  - **Geometric Priority** - $10 < Integer < 250$, used for intersecting objects
- Electric Parameters
  - **Incident wave direction** - absorber can be optimized if the wave direction is known

[2] where a is the skin depth $a = \frac{1}{\sqrt{\pi f \mu \sigma}}$

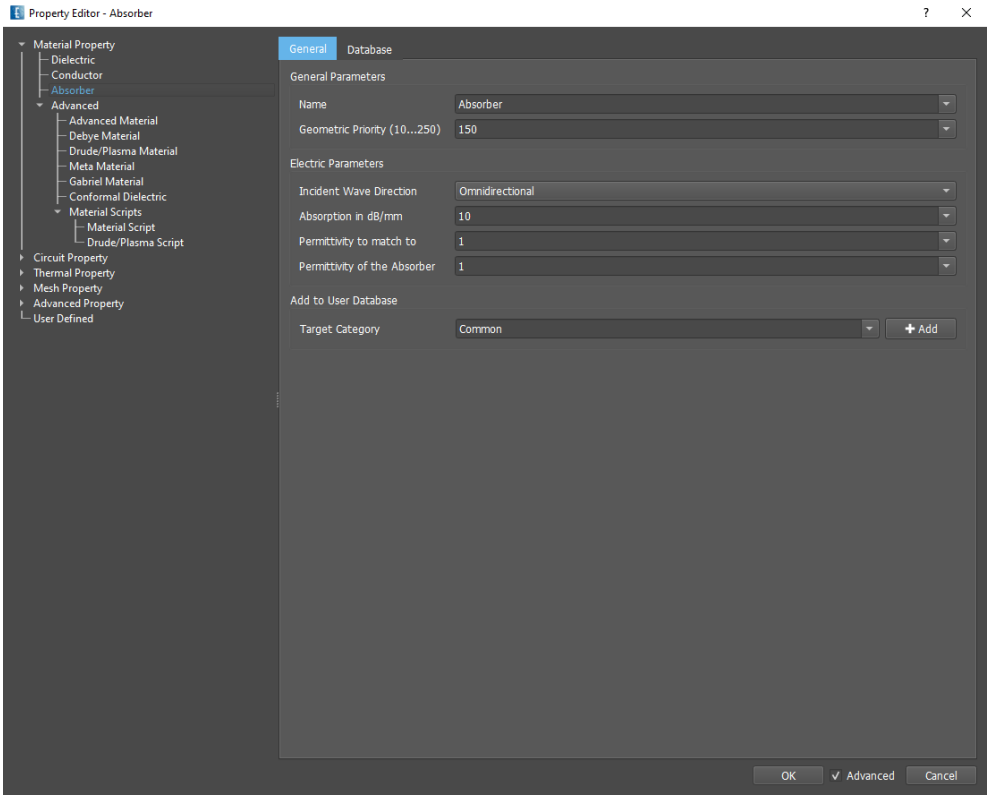


Figure 18.3.: Absorber property setup

  - **Permittivity to match** ε_1- ≥ 1 Surrounding material of the absorber, e.g. air or substrate
  - **Permittivity of the Absorber** ε_2- ≥ 1, e.g. ferrite permittivity
  - **Absorption** abs- ≥ 0, target absorption value in $\frac{dB}{mm}$
  - **Geometric Priority** - $10 < Integer < 250$, used for intersecting objects
- Add to User Database
  - **Target Category** - A unique name of an entry in the database list (editable to create new category)
  - **Add** the settings will be stored in a file called `.ganymed_wizmat.txt` in the local user directory and can be reused in other projects.

If a direction is given, the respective relative values are set to 1 and conductivities to 0.

18.2.4. Advanced - Material Object

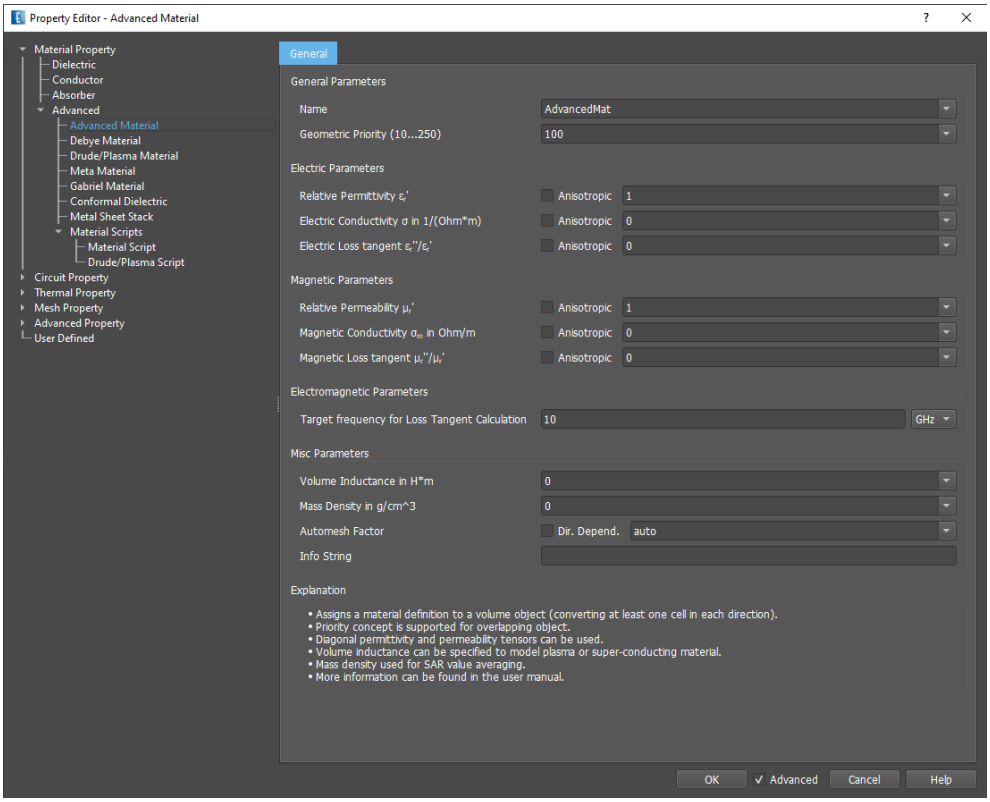


Figure 18.4.: Advanced Material property setup

Advanced Materials may be anisotropic, e.g the relative permittivity is a diagonal tensor

$$\underline{\underline{\varepsilon_r}} = \begin{pmatrix} 2 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 4 \end{pmatrix}$$

- General Parameters
  - **Name** - A unique name for display in Group

  - **Geometric Priority** - $10 < Integer < 250$, used for intersecting objects
- Electric Parameters
  - **Rel. Permittivity** - $\varepsilon_r \geq 1$
  - **Electric Conductivity** - $\sigma \geq 0$ in $\frac{1}{\Omega m}$
  - **Electric Loss Tangent** - Dissipation Factor $\tan\delta \geq 0$
- Magnetic Parameters
  - **Rel. Permeability** - $\mu_r \geq 1$
  - **Magnetic Conductivity** - $\sigma_m \geq 0$ in $\frac{\Omega}{m}$
  - **Magnetic Loss Tangent** - Dissipation Factor $\tan\delta_m \geq 0$
- Electromagnetic Parameters
  - **Target Frequency for Loss Tangent Calculation** for calculation of equivalent conductivity
- Misc Parameters
  - **Volume inductance** L_v. For special purposes an inductance $L = L_v \frac{\ell}{A}$ can be added to this material, where ℓ is the length and A is the cross section of the material box. E.g. for modelling a Plasma, a negative permittivity is often required. This can be modelled by setting $L_v = -\frac{1}{\varepsilon_0(\varepsilon_{Plasma}+1)\omega^2}$
  - **Mass density** has to be defined if Specific Absorption Rate should be evaluated with mass averaging, e.g. for 1g.
  - **Automesh Factor** Mesh refinement factor may be defined anisotropically
  - **Info String** for advanced usage

18.2.5. Advanced - Debye Material

A complex, frequency dependent permittivity and permeability can be defined using a **Debye** model:

$$\varepsilon(f) = \varepsilon_0\varepsilon_\infty + \sum_{\nu=1}^{n} \frac{\varepsilon_0\Delta\varepsilon_\nu}{1 + j\frac{f}{f_{el,\nu}}} - j\frac{\sigma}{2\pi f} \qquad \mu(f) = \mu_0\mu_\infty + \sum_{\nu=1}^{n} \frac{\mu_0\Delta\mu_\nu}{1 + j\frac{f}{f_{mag,\nu}}} - j\frac{\sigma_m}{2\pi f}$$

- ε_∞ is the permittivity for $f \to \infty$
- $f_{el,\nu}$ are the electric pole frequencies
- $\Delta\varepsilon_\nu$ are the relative changes of permittivity
- σ is an additional electric conductivity
- μ_∞ is the permeability for $f \to \infty$
- $f_{mag,\nu}$ are the magnetic pole frequencies
- $\Delta\mu_\nu$ are the relative changes of permeability
- σ_m is an additional magnetic conductivity
- n is the number of poles (jumps)

The above model parameters may be entered in the respective input fields **Electric Parameters** and **Magnetic Parameters** of the **Electric** and **Magnetic** Tabs. The **Info** Tab shows the frequency dependance of the created model.

- General Parameters

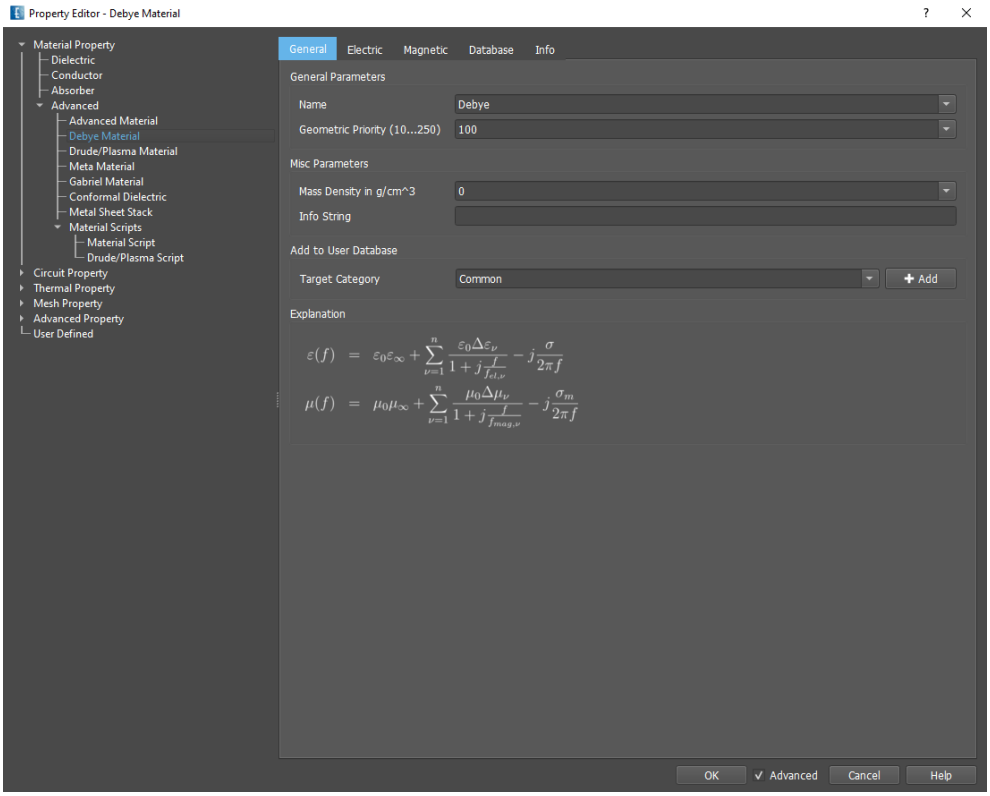


Figure 18.5.: Advanced Debye Material property setup

  - **Name** - A unique name for display in Group
  - **Geometric Priority** - $10 < Integer < 250$, used for intersecting objects
- Misc Parameters
  - **Mass density** has to be defined if Specific Absorption Rate should be evaluated with mass averaging, e.g. for 1g.
  - **Automesh Factor** Mesh refinement factor may be defined anisotropically
  - **Info String** for advanced usage
- Add to User Database
  - **Target Category** - A unique name of an entry in the database list (editable to create new category)
  - **Add** the settings will be stored in a file called `.ganymed_wizmat.txt` in the local user directory and can be reused in other projects.

In the the **Electric** and **Magnetic** Tabs calculators are available for the different conversions:

- **Model** → **Table**
  1. Enter Model Parameters (value at infinity, conductivtiy, list of jumps) in the upper input fields
  2. Enter **Number of Rows** for the table
  3. Click **Model→Table** to obtain a list of frequencies with the material dependence
- **Table** → **Model**
  1. Enter a list of frequencies (Hz) and material values (real and imaginary part) or click **Data-File→Table** and select a text file which contains these values
  2. Specify the **Number of Jumps** (Debye poles). (The number of jumps should be quite small. Much larger numbers do not generally increase the accuracy. It is good practice to start with 1 jump and increase only one by one. If the agreement does not improve other parameters can be tried, e.g conductivity, frequency range)
  3. Specify **Minimum** values at infinity
  4. Select **Use Conductivity** if it should be used for creating the model
  5. Click **Table→Model** Debye Model Parameters will be calculated by fitting curves to the entered frequency list and inserted in the upper input fields.
- **Data-File** → **Table** Import a given list to create the table
- **Table** → **Data-File** Save the table into a file

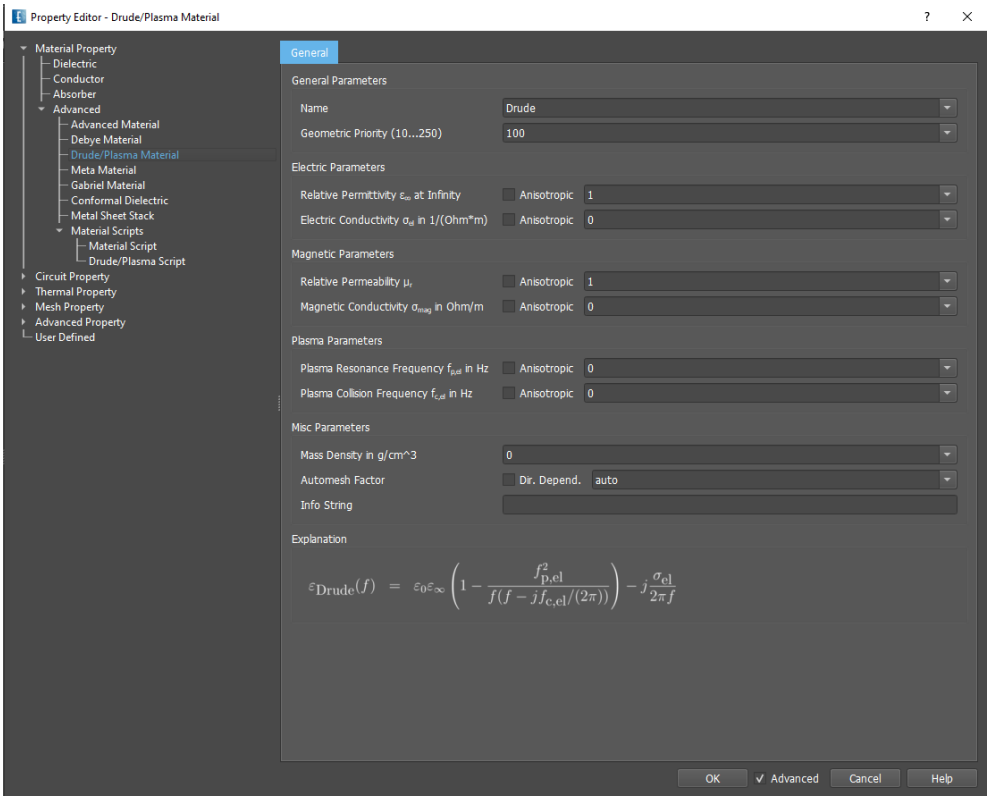


Figure 18.6.: Advanced Drude Material property setup

18.2.6. Advanced - Drude Plasma

A Drude Plasma is defined by electric and magnetic properties as well as plasma resonance frequency f_p and collision frequency f_c. The complex permittivity is calculated according to:

$$\varepsilon_{Drude}(f) = \varepsilon_0\varepsilon_\infty \left(1 - \frac{f_{p,el}^2}{f(f - jf_{c,el}/(2\pi))}\right) - j\frac{\sigma_{el}}{2\pi f}$$

Where f is the frequency and ε and σ are the permittivity and electric conductivity.

- General Parameters
  - **Name** - A unique name for display in Group
  - **Geometric Priority** - $10 < Integer < 250$, used for intersecting objects
- Electric Parameters
  - **Rel. Permittivity** - $\varepsilon_r \geq 1$
  - **Electric Conductivity** - $\sigma \geq 0$ in $\frac{1}{\Omega m}$
- Magnetic Parameters
  - **Rel. Permeability** - $\mu_r \geq 1$
  - **Magnetic Conductivity** - $\sigma_m \geq 0$ in $\frac{\Omega}{m}$
- Plasma Parameters
  - **Plasma Resonance Frequency**
  - **Plasma Collision Frequency**
- Misc Parameters
  - **Mass density** has to be defined if Specific Absorption Rate should be evaluated with mass averaging, e.g. for 1g.
  - **Automesh Factor** Mesh refinement factor may be defined anisotropically
  - **Info String** for advanced usage

18.2.7. Advanced - Meta Material

Meta materials are Materials with unusual and unconventional wave properties. One class of Meta materials has a negative refraction index (NRI). To model this NRI phenomenon, a Drude model for the electric field as well as for the magnetic field is implemented in EMPIRE XPU™. For frequencies lower than the electric resonant frequency $f_{p,el}$ and magnetic $f_{p,mag}$ resonant frequency an effective negative permittivity ε_{eff} as well as a negative permeability μ_{eff} can be observed. At the transition frequency the effective material parameters ε_{eff} and μ_{eff} are zero, which corresponds to an infinite wavelength at a non-zero frequency. Some values may be defined anisotropically in axes directions.

$$\varepsilon_{Meta}(f) = \varepsilon_0\varepsilon_\infty \left(1 - \frac{f_{p,el}^2}{f(f - j/(2\pi\tau_{el}))}\right) - j\frac{\sigma_{el}}{2\pi f} \quad \mu_{Meta}(f) = \mu_0\mu_\infty \left(1 - \frac{f_{p,mag}^2}{f(f - j/(2\pi\tau_{mag}))}\right) - j\frac{\sigma_{mag}}{2\pi f}$$

- General Parameters
  - **Name** - A unique name for display in Group

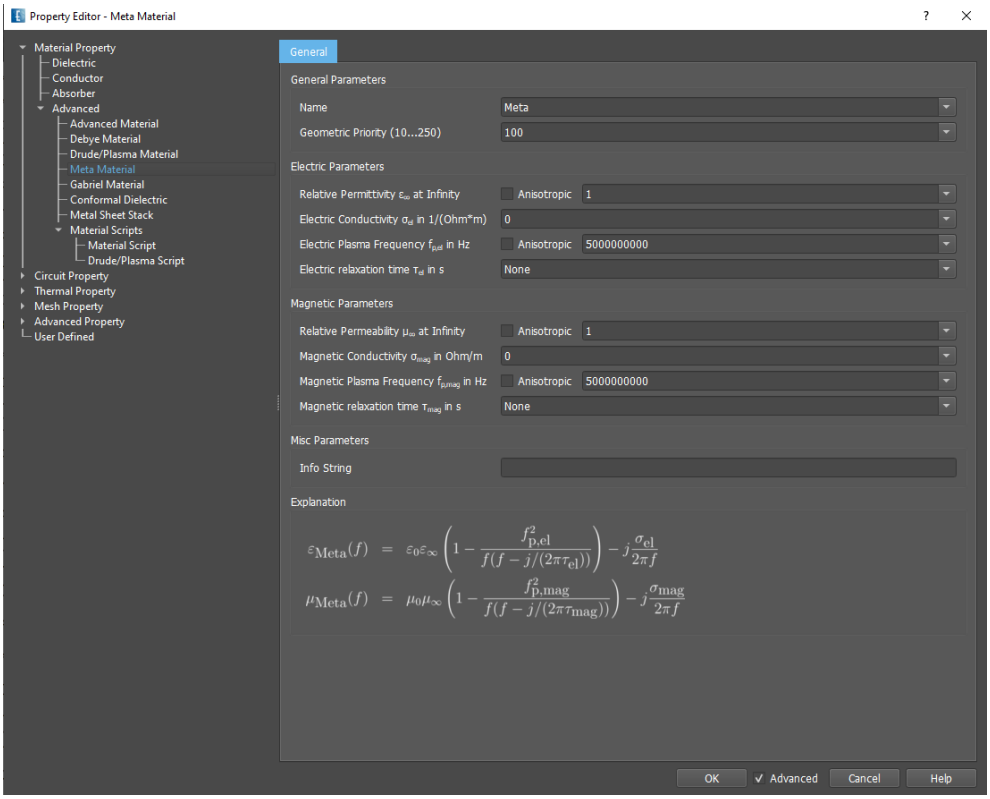


Figure 18.7.: Advanced Meta Material property setup

  - **Geometric Priority** - $10 < Integer < 250$, used for intersecting objects

- Electric Parameters
  - **Rel. Permittivity** - ε_r positive permittivity value, which corresponds to the effective permittivity for the frequency approaching infinity.
  - **Electric Conductivity** - Takes into account losses for the electric node in $\frac{\Omega}{m}$
  - **Electric Plasma Frequency** - Resonance frequency $f_{p,el}$
  - **Electric relaxation time** - Inverse of Plasma Collision Frequency τ_{el}
- Magnetic Parameters
  - **Rel. Permeability** - μ_r positive permeability value, which corresponds to the effective permeability for the frequency approaching infinity
  - **Magnetic Conductivity** - Takes into account losses for the magnetic node in $\frac{\Omega}{m}$
  - **Magnetic Plasma Frequency** - Resonance frequency $f_{p,mag}$
  - **Magnetic relaxation time** - Inverse of Plasma Collision Frequency τ_{mag}
- Misc Parameters
  - **Info String** for advanced usage

18.2.8. Advanced - Gabriel Material

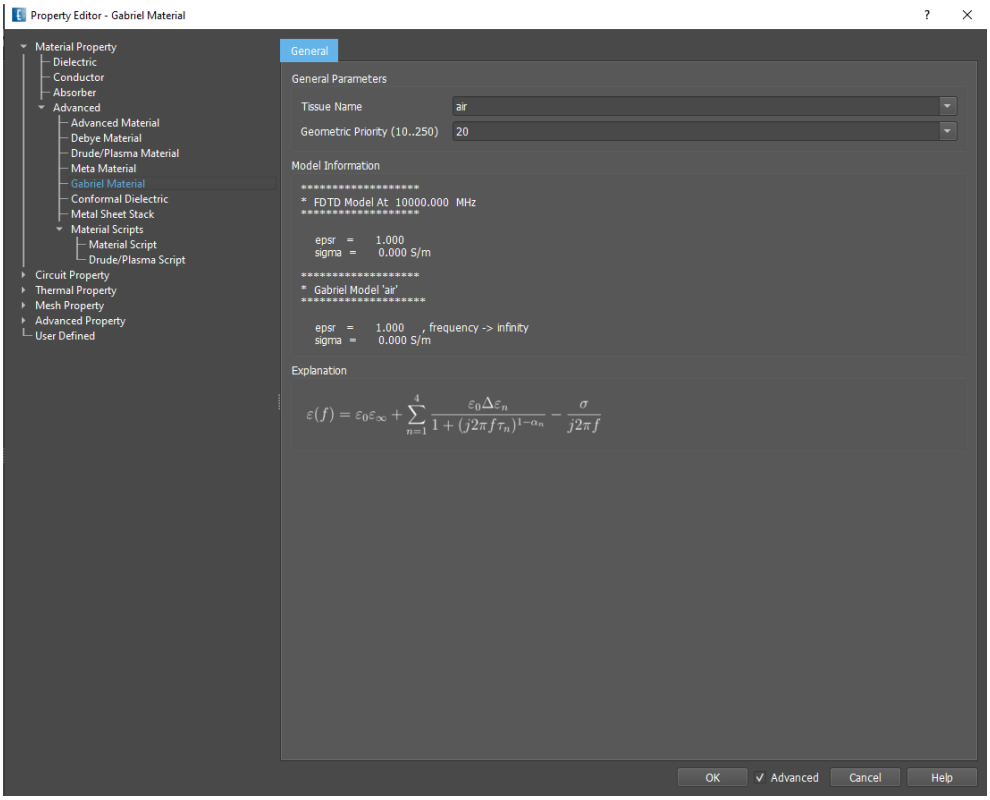


Figure 18.8.: Advanced Gabriel Material property setup

For modeling of tissues a collection of different materials can be selected from the internal database. The complex permittivity is obtained using the formula given in 46.20 which is valid for narrow band at the target frequency.

- General Parameters
  - **Tissue Name** - Select a given name form the drop down list
  - **Geometric Priority** - $10 < Integer < 250$, used for intersecting objects
- Model information
  - **FDTD Model** - The material values are displayed for the target frequency
  - **Gabriel Model** - The pole parameters of the Gabriel model is displayed for information

18.2.9. Advanced - Conformal Dielectrics

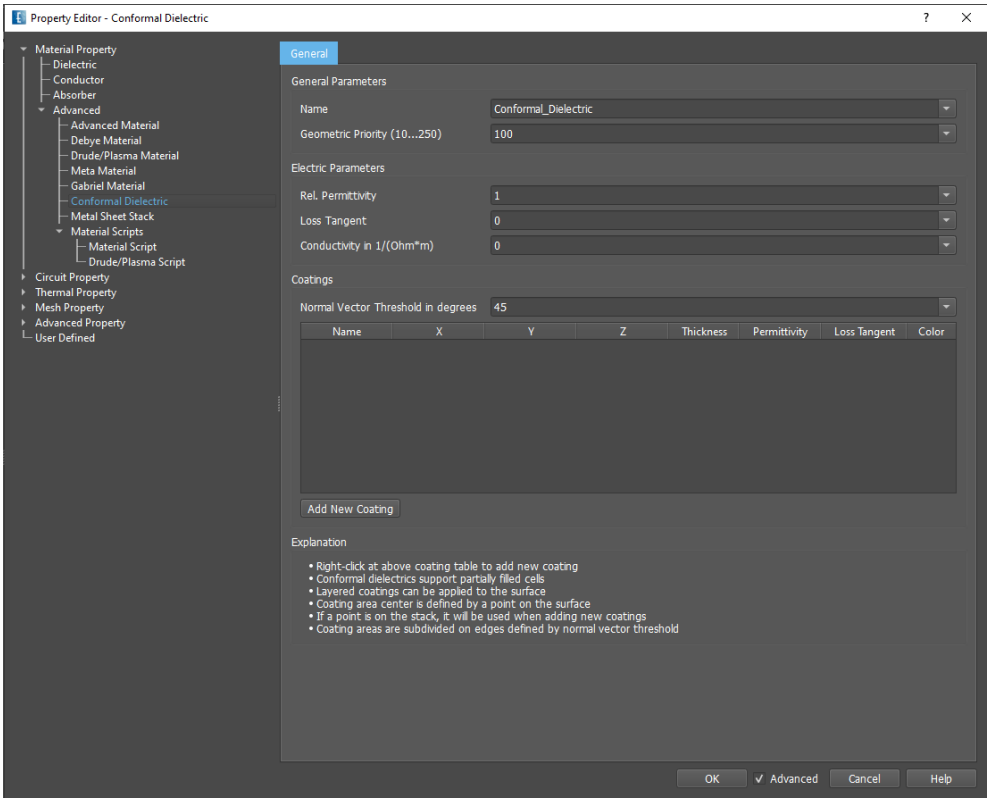


Figure 18.9.: Advanced Conformal Dielectric property setup

Conformal Dielectric property applies a specialized algorithm based on FEM (Finite Element Method) on the cells on the object surface:

- If the dielectric only partially fill the mesh cells weighted and directive node-correction factors are derived for each cell
- Layered coating can be applied to the surface with different permittivities and loss tangent factors
- Coating areas can be subdivided on edges (controlled by normal vector angle step)

Restrictions

- Conformal dielectric objects with different material properties may not intersect
- Thermal simulations are not supported

Properties

- General Parameters
  - **Name** - Name as displayed in the group
  - **Geometric Priority** - $10 < Integer < 250$, used for intersecting objects
- Electric Parameters (center body)
  - **Rel. Permittivity** - $\varepsilon_r \geq 1$
  - **Electric Loss Tangent** - Dissipation Factor $\tan\delta \geq 0$
  - **Conductivity** - $\sigma \geq 0$ in $\frac{1}{\Omega m}$
- Coatings (Dielectric layers from center body to outside)
  - **Normal Vector Threshold** $\alpha \geq 0$ in degrees. Angle step used to identify piecewise smooth surfaces. Together with the "Reference Point" on the surface it can be used, e.g. to separate inside and outside of a shell.
  - **Name** of the dielectric layer for identification purposes
  - **x y z** Reference point on the surface
  - **Thickness** - $t \geq 0$ value in current units (may be smaller than cell spacing)
  - **Permittivity** - $\varepsilon_r \geq 1$ Dielectric constant of the sheet
  - **Loss Tangent** - Dissipation Factor $\tan\delta \geq 0$ of the sheet
  - **Buttons** (available after **Add New Coating**)
    - Color of the layer (displayed in 3D Results mode)
    - Copy layer
    - Pick reference point (to be used in 3D Design mode)
    - Delete layer
  - **Add New Coating** A new sheet will be added to the list. Added sheets can be duplicated or deleted from the list. Using Drag & Drop the layer order can be changed

Hint: Application Note

More information is available in Chapter 29.2.4

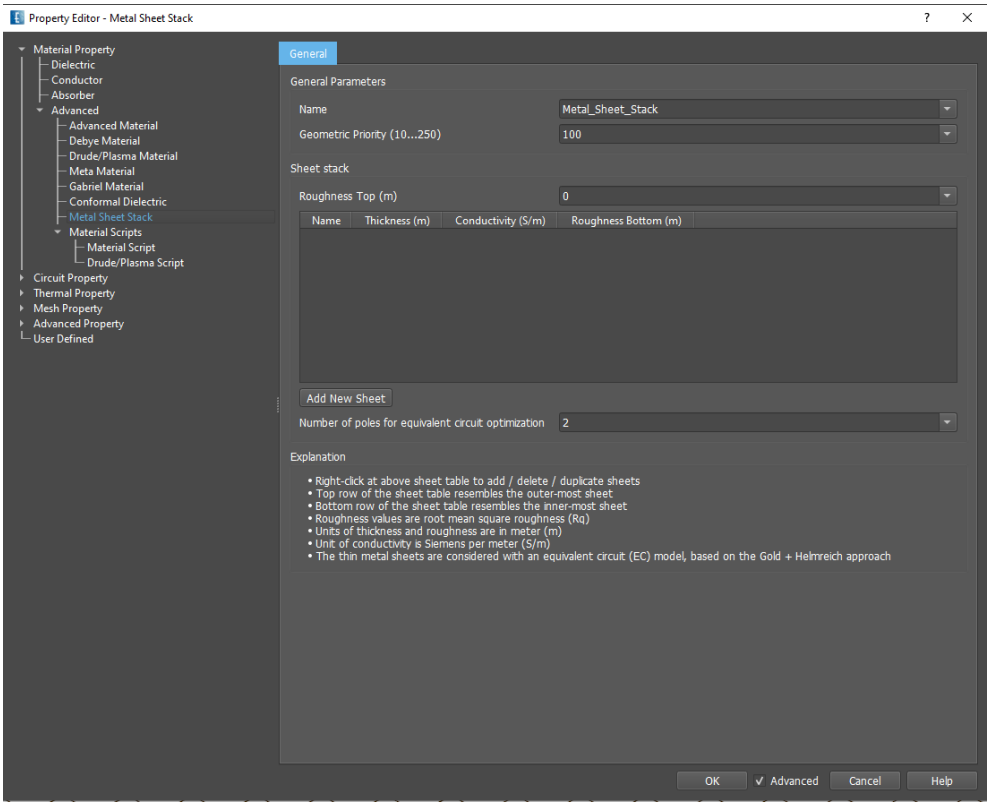


Figure 18.10.: Advanced Metal Sheet Stack property setup

18.2.10. Advanced - Metal Sheet Stack

The metal sheet stack can be used to define layers of metal sheets. Depending on their thicknesses, roughnesses and conductivities an equivalent circuit is derived for the conductor model and inlcuded in the simulation. For the roughness a Gold + Helmreich approach is applied.

Properties

- General Parameters
  - **Name** - Name as displayed in the group
  - **Geometric Priority** - $10 < Integer < 250$, used for intersecting objects
- Sheet Stack
  - **Roughness Top** - Value for top side in m
  - **Name** of the layer
  - **Thickness** of the metal sheet in m
  - **Conductivity** $\sigma \geq 0$ in $\frac{1}{\Omega m}$
  - **Roughness Bottom** Value for bottom side in m
  - **Add New Sheet** A new sheet will be added to the list. Added sheets can be duplicated or deleted from the list.
- Number of poles for EC optimization - $1 \leq value \leq 5$

18.2.11. Advanced - Material Scripts

A script can be used for material definition, e.g. to model a functional material distribution. An example can be found in Chapter 28.2

18.3. Circuit Property

A circuit property can be assigned to objects and be used in the EM simulation. These objects are distributed over the volume, surface or line.

18.3.1. Volume objects

- The object type has to be a box which can be flat or thin and has to be defined between two conductors
- The direction of current flow has to be specified
- Metal caps ensure a homogeneous current flow
- **Resistor** - Value (> 0) has to be entered in Ω.
- **Capacitor** - Value (> 0) has to be entered in Farad. [3]
- **Inductor** - Value (> 0) has to be entered in Henry. [4]
- **LRC circuit** - Parallel circuit of **Rp**, **Cp**, and **Rs-Ls**. Values (> 0) or **None** have to be entered in Henry, Ohm or Farad.

[3] Maximum value can be determined from $N_{timesteps}\delta t > 3\tau = 3RC$
[4] Maximum value can be determined from $N_{timesteps}\delta t > 3\tau = 3\frac{L}{R}$

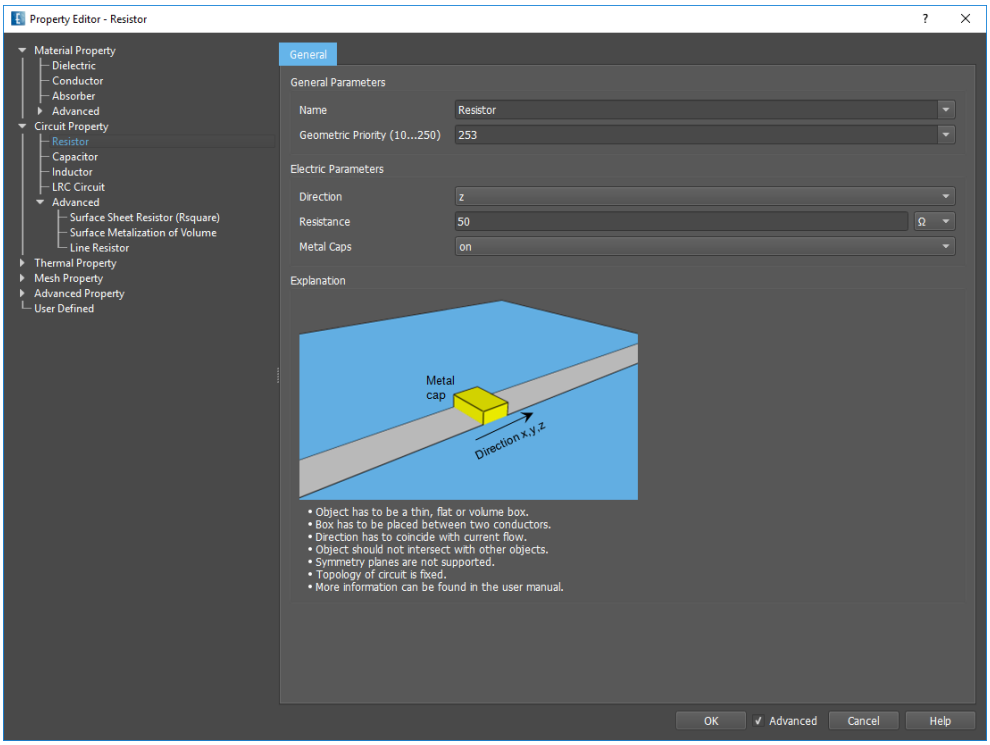


Figure 18.11.: Lumped resistor definition

18.3.2. Advanced - Surface objects

Surface Sheet Resistor (Rsquare)

A direction-less square resistance can be entered for a box object if current direction is not known

- A sheet resistance in Ω can be defined
- The box must be thin, e.g. cover only one grid line in one direction (if the thickness is larger than a cell both top and bottom get the Rsquare property)
- The priority number can be used if the object intersects others

If the box has a width w and a length l and the current is flowing in length direction the resistance is given by $R = R_{square}\frac{l}{w}$

Surface Metallization of Volume

Surface metalization of volume objects (solids).

- **Metallization** - Surface will be metalized with ideal conductor or R_{square} resistance.
- **Mask** - Metallization is switched off where **Mask** objects intersect **Metallization** object.
- **Unmask** - Cancel **Mask** definition where **Unmask** objects intersect **Metallization** object.

18.3.3. Advanced - Line objects

Line Resistor

- A line resistor with a value in $\frac{\Omega}{m}$ can be defined
- The object type has to be a line object (line polygon).

18.4. Thermal Property

For thermal simulation additional objects may be needed which are ignored in EM simulations.

- **Heat Sink** - Defines an area with a defined resistance to the ambient temperature. Instead of entering a complex heat sink geometry, just a box is needed for the definition
- **Heat Source** - Applies on box geometries as an alternative definition for heat sources as library objects
- **Constant Temperature** - Sets a constant temperature to the objects in this group
- **Convection Area** - If convection is set to manual, this box defines a convection area. The direction of the heat transfer and an alternative ambient temperature may be defined.
- **Radiation Area** - If radiation is set to manual, this box defines a radiation area. The direction of the heat transfer and an alternative ambient temperature may be defined.

18.5. Mesh Property

18.5.1. Mesh Hint

With this property it is possible to assign local meshing hints to objects. This can be used for mesh refinement of certain areas or to force the cell spacing not to become too small. This property acts on bounding boxes of objects. If this attribute is assigned to an object with no further properties this object is ignored for simulation. This attribute can also be added to existing properties. If several mesh hints should be used for overlapping areas a priority number defines the dominating mesh hint.

- **Geometric Priority** Integer value specifying priority of the mesh hint
- **Strictness** If set to **force** the contstraint will be fulfilled strictly.
- **Edge Use** If switched to **On** grid lines will be generated on the bounding box edges
- **Edge Max. Interval** Inserts grid lines at the edges with a specified spacing
  - `auto` the spacing at the edge will be defined by automeshing algorithm
  - `100` a spacing of 100 units will be used at the edges by inserting 3 grid lines
  - `100@5` a spacing of 100 units will be used at the edges by inserting 5 grid lines
  - `100,200` creates grid lines with a spacing of 100 (200) at minimum (maximum) edge
  - **Apply to** Use **Edge Max. Interval** for all edges or on bounding box edges, only
- **Refinement** Relative refinement factor, applies a subdivision of the mesh covered by the group objects by the factor given
- **Max. Interval** Absolute refinement number, all grid lines in this area get a maximum spacing of the entered value.

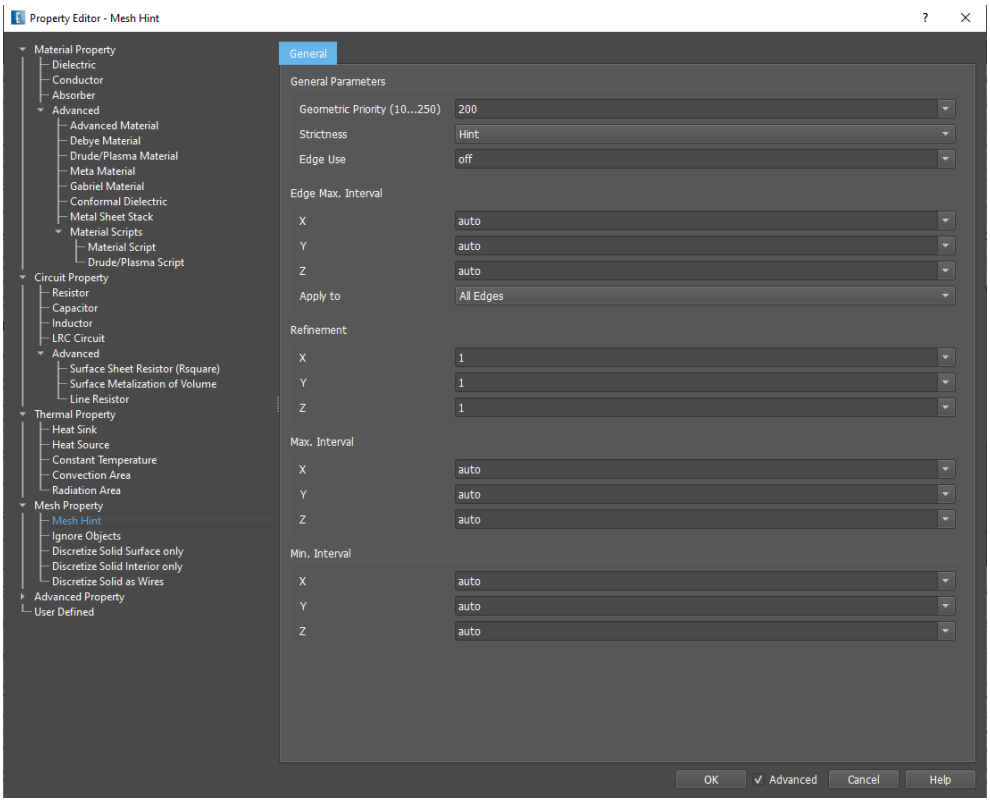


Figure 18.12.: Mesh hint definition

- **Min. Interval** Absolute number, preventing cell spacing lower than this limit

18.5.2. Mesh Accuracy and Quality Check

If this property is set to a group the objects on it define areas where the hard mesh refinement will be restricted to. Restrictions can be applied to different directions, only. See Section 24.8 for details.

18.5.3. Ignore Object

If this property is set to a group the objects on it will be ignored for simulation. If there are further properties set to the group they will be ignored.

18.5.4. Discretize Solid Surface only

surfdisc - Used as additional property for continuous meshing of thin solids. If the thickness of a solid is smaller than the cell spacing, solids break into pieces and holes or gaps can occur after meshing. To avoid this this property ensures that the solid will be meshed continuously.

18.5.5. Discretize Solid Interior only

bulkdisc - Used as additional property for meshing of solids where possible thin parts will be ignored if the cell size is larger than the thickness. If the thickness of a solid is smaller than the cell spacing, solids break into pieces and holes or gaps can occur after meshing.

18.5.6. Discretize Solids as wires

wiredisc - Used as additional property for continuous meshing of thin wires.

18.6. Advanced Property

18.6.1. Voxel model

A meshed body model can be included into the simulation to evaluate the field in tissues, e.g. ACD, SAR, EIAV.

- **General**
  - **Name of the voxel file** (ending **.vxx** or **.zip**), must be located in project folder
  - **Geometric Priority** Used in case of intersection with other objects
  - **Gabriel target frequency** used for determination of Gabriel material parameters (ε_r and σ_e)
  - **Frequency of field dump** Can be set to **auto** if only one frequency is selected for storage
- **Display**
  - **2D display plane in % of voxbox** Slice of the voxel model to be displayed in 2D Design mode
  - **2D display** By default coordinates of the slice are displayed near the anchor, can be disabled
  - **3D display** Either whole or plane (slice) display of the voxel model in 3D mode

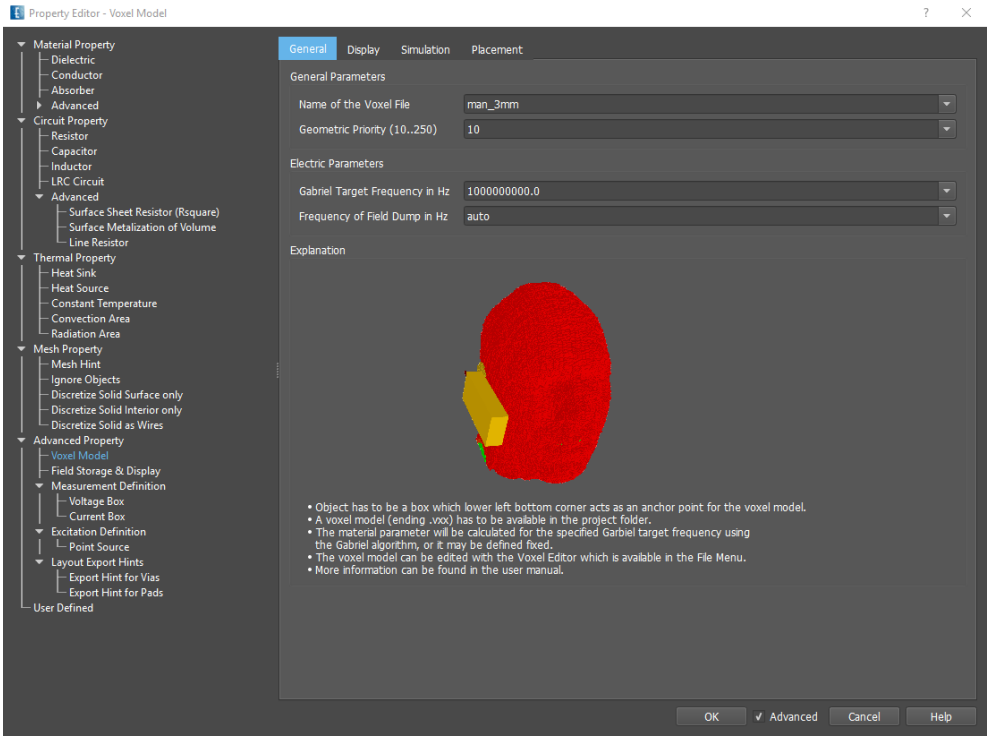


Figure 18.13.: Advanced property definition

  - **Min Permittivity for display** Tissues with lower values will be hidden
  - **Min Mass Density for display** Tissues with lower values will be hidden
  - **Min Conductivity for display** Tissues with lower values will be hidden
  - **Tissue(s) to display (list of numbers)** Number separated by commas, see voxel editor for tissue numbering
- **Simulation**
  - **Fixed Permittivity** if value can be neglected for simulation
  - **Fixed Conductivity** if value can be neglected for simulation
  - **Store Voxel Info** on / off
  - **Min Mass density for simulation** Tissues below this threshold will be disregarded for simulation
- **Placement**
  - **Clip Region** used if only part of the model is used
  - **Erase Start Points** is used if only part of the model is used
  - **Anchor** defines insertion point relative to voxel box coordinates x0, y0, z0
  - **Rotation** Limited to multiples of 90 degrees

18.6.2. Field Storage & Display

Near and far field recording is done by defining Field Monitors. For compatibility reasons field recordings and animations can also be defined in groups with storage and display properties. Please refer to chapter 19 for the meaning of the different options.

18.6.3. Measurement Definition

Voltage Box

- A voltage box defines the integration path for the electric field $U = \Sigma E \Delta s$
- The direction of the integration path is determined by the largest box dimension
- In the direction of the integration path the box has to cover at least 2 grid lines
- In the perpendicular direction the voltage box has to cover exactly one grid line
- For each port a unique number has to be defined.

Current Box

- A current box fixes the integration path for the magnetic field $I = \Sigma H \Delta s$
- The direction of the current flow has to coincide with the smallest box dimension
- The box has to be placed between exactly 2 grid lines in current flow direction
- The perpendicular cross section has to cover the the conductor which current should be evaluated.
- For each port a unique number has to be defined.

18.6.4. Excitation Definition

Point Source

Hard or soft excitation of an electric field inside a box volume

- **Port Number** Determines sub folder name
- **Direction of Excitation** Electric field component to be excited inside the box volume
- **Source Type** Hard - or soft excitation
- **Weight** Scale factor
- **Delay in s** Time shift of pulse
- **Reference for field dumps** Normalization
- **Function** Pulse shape

18.6.5. Layout Export hints

Only needed if data should be exchanged with PCAD program.

- **Export Hints for Vias** - Group containing vias
- **Export Hints for Pads/Ports** - Group containing pads or ports
- **Export Hints for Layers** - Used for mapping layer names with **Altium** naming convention

18.6.6. No Dispersion Correction

Excluding groups and objects from directive dispersion correction defined in **EM Setup- Enhancement**, see section 23.2.5. If the volume around such structures needs to be excluded from the correction apply this special property to the groups of the volume.

18.6.7. Experimental

Used mainly for debugging purposes or academic experiments.

18.6.8. User defined

All properties which are defined in the dedicated menus above are mapped into text strings. Used for compatibility reasons or special purposes, e.g. to copy and paste properties from one file to another or to use in script files.

19. Field Monitors

Near and far field recording and display are done by defining **Field Monitors**. The button **Create Field Monitor** or a right click on the respective entry in the side menu can be used. Different field monitors are available for near and far field as well as field probes and temperature distribution. Storage properties have to be defined before the simulation. After simulation the results will be stored in files called, e.g. `sub-1/Emvolume.dbx` (near field) or `sub-1/Farfield_1` (far field) or `sub-1/th.dat` (temperature). Field display properties can be defined before or after the simulation.

Hint: Copy Field Monitor

In order to copy a Field Monitor, open the **Field Monitors** list on the side menu, click right mouse button on the respective Field Monitor, click on copy button

Hint: Delete Field Monitor

In order to delete a Field Monitor, open the **Field Monitors** list on the side menu, click right mouse button on the respective Field Monitor and select **Delete**

19.1. EM Field

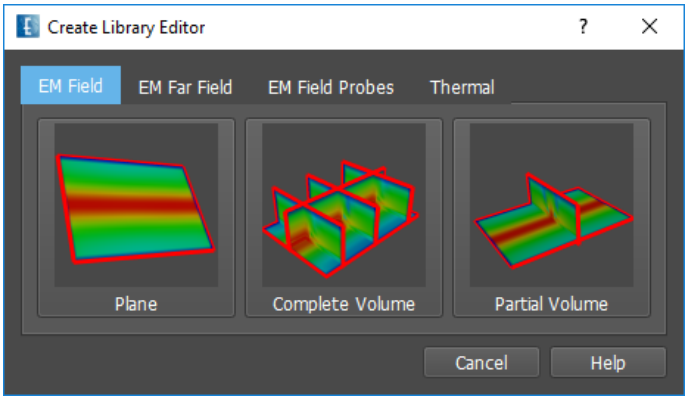


Figure 19.1.: Field Monitor EM Field

With **EM Field** near fields are recorded in a defined area and can be visualized in the **3D Results** Tab together with the structure.

- **Planar EM Field** Specifies a plane for field storage and display. The position will be snapped to the nearest fitting mesh line. Displays E or H field, only.
- **Complete EM Field** Specifies the complete simulation domain for field storage (memory intensive). For display several planes may be defined and varied with a slider.

- **Partial EM Field** Defines a volume which can be limited by the user. For display several planes may be defined and varied with a slider.

Each **EM Field** contains **Parameters** for the setup of storage and display.

Hint: Field Plane Display

The field is displayed on a mesh plane which is nearest to the specified plane in the navigation tree on the left.

Hint: Surface Display

In case of a volume field monitor the field can also be displayed on a curved surface which is part of the volume. For this the surface object can be selected with a right click on the field monitor in the navigation tree and use **Pick object as Render Surface**.

19.1.1. Storage Options

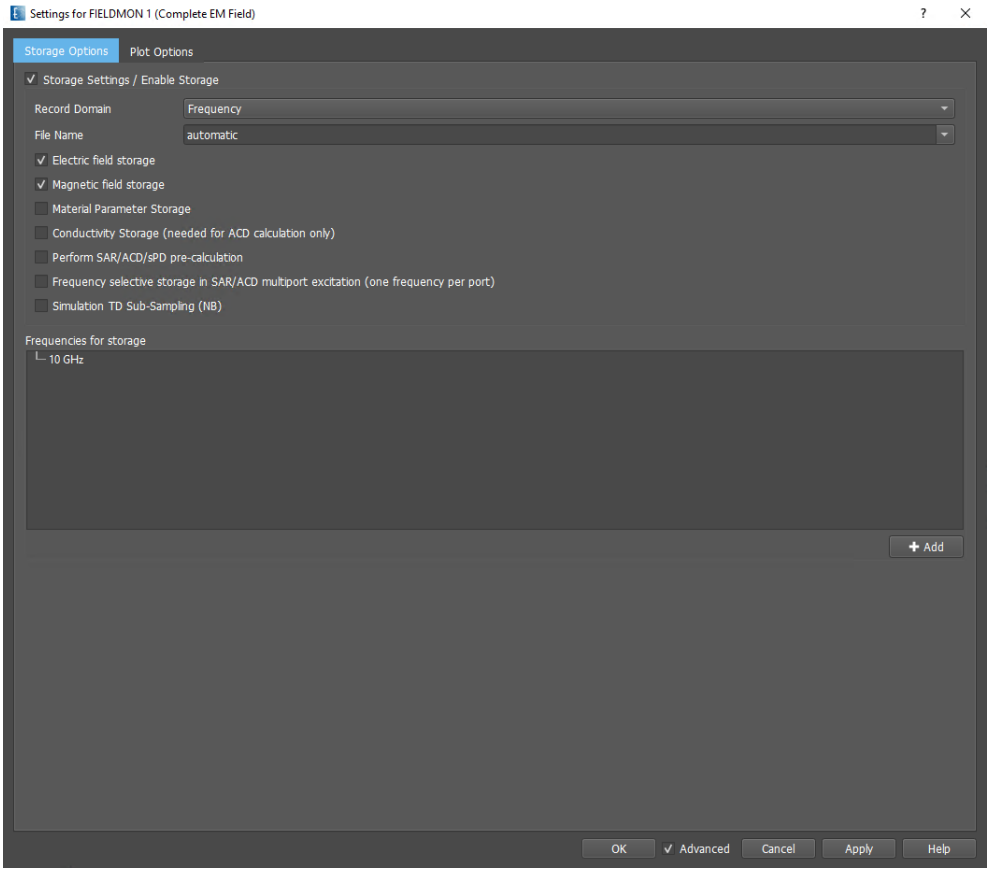


Figure 19.2.: Field dump storage definition

Field dumps can be defined for both time and frequency domain. In both cases field dumps will be generated at the end of the simulation. Files with ending `.dbx` are generated and can be visualized in the **3D Results** tab.

Frequency domain dumps

Frequency dumps are generated by an on-line DFT during the simulation run and save the complex valued E and H field. The performance can slow down and the memory consumption can be high if complete volume is chosen or many frequency points are used.

General Settings

- ☐ **Storage Setting / Enable Storage** If disabled no field dump storage, for additional display only
- **Record Domain** Frequency or Time domain
- ☐ **Electric field storage** Stores all electric field nodes covered by the volume
- ☐ **Magnetic field storage** Stores all magnetic field nodes covered by the volume plus one additional plane for averaging purposes
- ☐ **Conductivity storage** Stores electric conductivity which is covered by the volume for ACD calculation.
- ☐ **Simulation TD-Sub-Sampling (NB)** If enabled, simulation speed can be improved for narrow band excitations
- **Frequencies for Storage** Specify frequencies for recording.

Advanced Settings

- **File Name** Is set automatically by default, but can be specified for compatibility reasons or special purposes
- ☐ **Material Parameter Storage** If enabled, information is stored for display of material distributions.
- ☐ **Perform SAR/ACD/sPD pre calculation** A cache file will be created in postprocessing with the settings defined in 3D Plot options
- ☐ **Frequency selective storage in SAR/ACD multiport excitations** Useful for multi-frequency SAR/ACD evaluation

Time domain dumps

Time domain dumps save the real valued E and H field at certain time steps.

If the time steps should be saved for fixed values the **Fast Time Domain Storage Algorithm** has to be switched off and the performance will slow down considerably. In this case only one file with ending `.dbx` is generated which contains all time steps.

If it is sufficient to store the field at approximate intervals the high speed algorithm (multiple time stepping) can be maintained by switching on the Fast Time Domain Storage Algorithm. In this case several files with ending `.dbx` are generated for each suitable time step.

General Settings

- ☐ **Storage Setting / Enable Storage** If disabled no field dump storage, for additional display only
- **Record Domain** Frequency or Time domain
- ☐ **Electric field storage** Stores all electric field nodes covered by the volume
- ☐ **Magnetic field storage** Stores all magnetic field nodes covered by the volume plus one additional plane for averaging purposes
- ☐ **Fast Time Domain storage algorithm** Decides whether approximate intervals or exact time steps are used
- **Time Domain approximate interval** Creates time domain fields compatible with high speed algorithm approximately every entered value time step (visible if FTD Algorithm is enabled)
- ☐ **Simulation TD-Sub-Sampling (NB)** If enabled, simulation speed can be improved for narrow band excitations

- **Timestep for Storage** Creates data at exact time steps (slow algorithm, only. It is possible to define equidistant sequences or single points (only usable if FTD Algorithm is disabled)

Advanced Settings

- **File Name** Is set automatically by default, but can be specified for compatibility reasons or special purposes

☐ **Material Parameter storage** Saves material distribution for display

Hint: Field dump box size

A field dump requires approx. $NF \times Nx \times Ny \times Nz \times 48Bytes$ or $NT \times Nx \times Ny \times Nz \times 24Bytes$ of memory where NF is the number of frequency points NT is the number of time steps and Nx, Ny, Nz are the number of cells enclosed by the dump box. In order to save memory it is advantageous to keep the numbers as low as possible.

19.1.2. 3D Plot Options

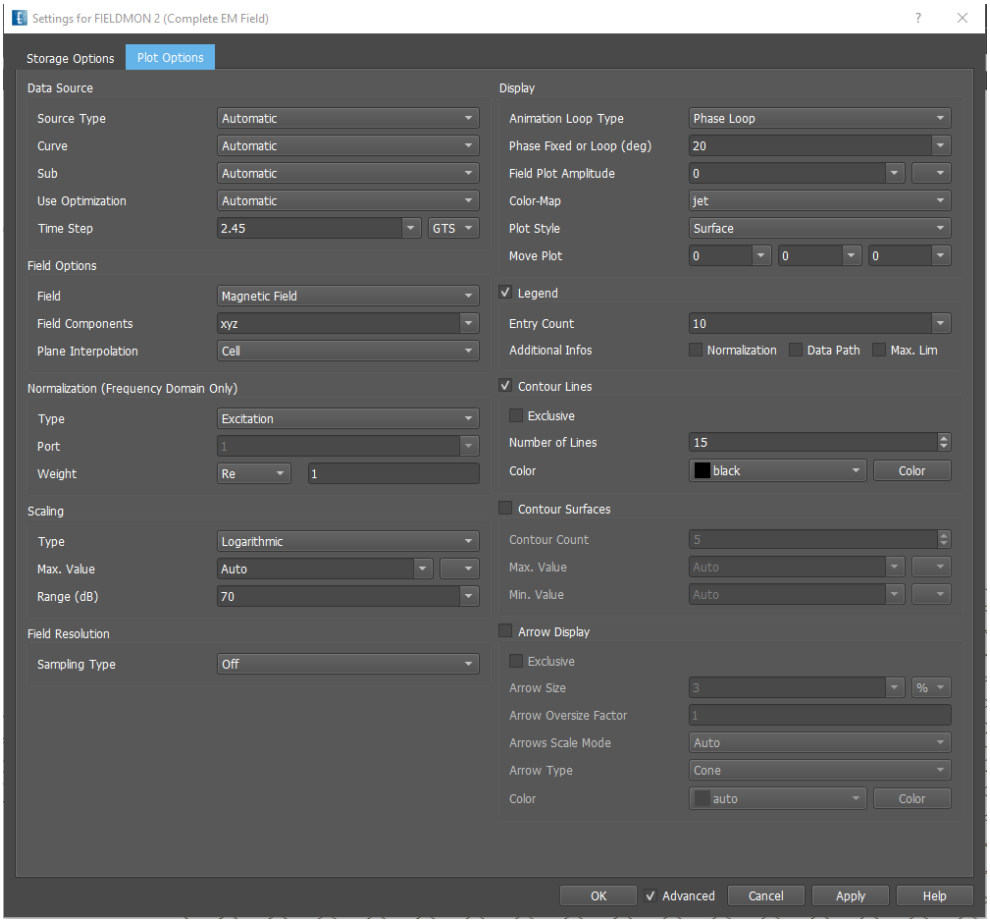


Figure 19.3.: Field animation definition

The following settings can be entered before or after the simulation.

Data Source

- **Source Type**
  - **Automatic** Search a suitable near field file and optionally specify name or result folder
    - **Curve** Location and name of the source file
    - **Sub** If multiple ports are excited the respective number can be selected
    - **Use Optimization** If variables have been swept the respective optimization folder can be selected
  - **Manual** Select the near field file name `.dbx` by browsing result folders.
  - **Superposition** Select multiple near field files and set up a table for excitation
  - **Superposition (Circuit)** Use results of a circuit simulation for set up a superposition of near field files
- **Frequency/Time selection** - If more than one frequency or time step has been defined for the field dump, it has to be specified here

Field Options

- **Field:** (In case of Planar field dump only E and H fields can be displayed)
  - Electric Field **E** in V/m
  - Magnetic Field **H** in A/m
  - Magnetic Flux Density **B** - calculated from $\mu_0 H$ in Vs/m^2
  - Current Density **J** - calculated from $\nabla \times \vec{H}$ in A/m^2
  - Surface Current Density **j** - calculated from $\hat{n} \times \vec{H}$ in A/m
  - Power Flow **S** - calculated from $\vec{E} \times \vec{H}$ in W/m^2
  - spatial-averaged Power Density (sPD) **S** - area averaged calculated from $\vec{E} \times \vec{H}$ in W/m^2 according to IEEE 63195-2.
  - Power Loss Density **S** - calculated from $\vec{E} \cdot \vec{J}$ in W/m^2
  - Specific Absorption Rate **SAR** - mass averaged absorbed power in tissues - calculated from $\sigma |E|^2/\rho$ in W/kg
  - Internal Averaged Electric Field **EIAV** - cube averaged of electric field in tissues in V/m
  - Averaged Current Density **ACD** - area averaged current density in tissues - calculated from $\sigma \vec{E}$ in A/m^2
  - Rel. Permittivity $\boldsymbol{\varepsilon_r}$ - permittivity distribution if Material Storage is enabled
  - Conductivity $\boldsymbol{\sigma}$ - conductivity distribution if Material Storage is enabled in $1/(\Omega m)$
- **Field Components**
  - **xyz** all components are considered for display
  - **xy** one component omitted for display
  - **x** only one component considered for display
  - **perpendicular** display of component which is normal to surface
  - **in-plane** display of components which are tangential to surface

  - **LHCP** left-handed circular component with specified direction, e.g. $H_{z+}^{LHCP} = H_x - jH_y$
  - **RHCP** right-handed circular component with specified direction, e.g. $H_{z+}^{RHCP} = H_x + jH_y$
- **Plane Interpolation** Specifies the averaging in cells or nodes
  - **Cell Interpolation** Default setting and valid also for inhomogeneous material distributions.
  - **Node Interpolation** Field values will be interpolated in node direction (along cell edges). Because of possible material discontinuities between cells this should only be applied in homogeneous areas.

Normalization

Settings apply to frequency domain result, only.

- **Type** - Field values can be normalized for different frequency domain results, e.g. **Current, Voltage, Power** in order to obtain fields at (1A, 1V, 1W) source input
  - **Excitation** is used by default. Field values are referenced to the excitation function **ef**. Recommended for general source, e.g. plane wave.
  - **Voltage** is used if the field values should be normalized to a source which applies 1V to the structure.
  - **Voltage (Incoming)** is used if the field values should be normalized to a source with 1V available voltage.
  - **Current** is used if the field values should be normalized to a source which feeds 1A into the structure.
  - **Current (Incoming)** is used if the field values should be normalized to a source with 1A available current.
  - **Power** is used if the field values should be normalized to a source which feeds 1W into the structure.
  - **Power (Incoming)** is used if the field values should be normalized to a source with 1W available power.
  - **All Ports Power** is used if the field values should be normalized to all sources which feeds 1W into the structure.
  - **All Ports Power (Incoming)** is used if the field values should be normalized to all sources with 1W available power.
- **Port** Specify the port number if multiple ports are present
- **Weight** Scaling factor

Scaling

- **Type** Linear or logarithmic scaling
- **Max Value** If set to automatic, the maximum found in field dump is used
- **Range (dB)** Dynamic range. Effects on color map, legend and equi-lines

Field Resolution

This option determines the resolution for field display, e.g. to accelerate the animation for huge field dumps

- **Off** Use all values of the field dump for display
- **Equidistant Sampling** Use equidistant support points for display
- **Min. Interval** Neglect cells below a minimum value
- **Sub Sampling** Use only every nth cell for display where n is the sample number

Each item can be specified anisotropically.

Display

- **Animation Loop Type** - Sweep (Phase Loop/Freq/Time) or no-sweep (Off, Real, Imag, Fixed Phase). If set to **Off** or **Freq/Time** the magnitude of the selected components are displayed.
- **Field plot amplitude** Hill plot display with maxmimum height determined by this value.
- **Color-Map** List of predefined color gradients
- **Plot style** Surface, wireframes or points
- **Move Plot** Shift coordinates used for display

☐ Legend

- **Entry Count** - Switch on by setting number of keys for field strength vs. colors. A special syntax allows the usage of specific colors for field value ranges: `val1:col1_val2:col2_val3:col3` where color **col1** is used for field values greater equal **val1** and so on.
- **Additional Infos** Checkmarks for extended legend

☐ Contour lines

if enabled lines of constant field strength are displayed

- ☐ **Exclusive** If enabled, only contour lines are displayed
- **Number of lines** Number of contour lines (depending on values defined in Scaling)
- **Color** Color of lines from list or custom color. **Field** applies line colors according to the field strength.

☐ Contour Surfaces

if enabled surfaces of constant field strength are displayed. (Use cutting tool for displaying interior surfaces)

- **Contour Count** Number of surfaces
- **Max value** Maximum field strength for surface
- **Min value** Minimum field strength for surface

☐ Arrow Display

if enabled arrows representing direction and strength of the field are displayed

☐ **Exclusive** Vector display only

- **Arrow Size** Length of maximum value arrow in units
- **Arrow oversize Factor** Vector magnification
- **Arrows Scale Mode** If enabled vector length depends on field strength
- **Arrow Type** Cone, arrow, line
- **Color** Coloring according to field strength or single color

Hint

Arrows can only be displayed for selected items in **Animation Loop Type**: real_part, imag_part, fixed_phase or phase_loop.

SAR/EIAV/ACD/sPD

If the **Field** is selected to SAR/ACD/EIAV/sPD in the **Field Options** some options are disabled and some other are enabled:

- **SAR** Specific Absoprtion Rate
  - **SAR Averaging Mass** The averaging mass has to be specified. If no averaging should be used it can be switched off
  - **Cube interpolation** Different standards of interpolation. Recent standard is **ieee/62704**
- **EIAV** Internal Averaged Electric Field
  - **EIAV Av. Cube Edge** Size of averaging cube
  - **Cube interpolation** Different standards of interpolation. Recent standard is **ieee/62704**
- **ACD** Averaged Current Density
  - **Field Components** Choose direction of field
  - **ACD Averaging** Size of averaging area
  - **Cube interpolation** Whole or partial
- **sPD** Spatial Averaged Power Density
  - **sPD Averaging Area** Size of averaging area
  - **sPD Averaging Grid Size** This value sets the grid size "_dT_" of the equidistant mesh for power density averaging with the algorithm described in IEC/IEEE 63195-2 Annex D.1.3. Smaller values lead to more accurate results, but also longer computation times. If set to "Auto", the minimum of 1mm and lambda/10 is applied, which is the value suggested in the standard.
  - **sPD Averaging Definition** IEEE norm selection

Hint: Field monitor for derived quantities

In order to evaluate current densities or SAR/EIAV/ACD/sPD a Complete or Partial EM Field has to be used.

19.1.3. 3D Result Options

The following options are available in the 3D Results Tab with a right click on the field display

Add Marker

- Right click on field display adds a marker at cursor position. Multiple markers can be set
- Left click on marker and drag shifts the position of the marker
- The markers including position are listed in the field monitor in the respective display plane
- The marker entries can be edited, removed or moved to maximum in the list
- Marker font size, color and **Show Position** option can be set in the **3D Result Options** 

FIELDMON

- **Edit** Open the **Plot Options** of this monitor
- **Hide** Disable display of this monizot
- **Transparency** Set opaque value
- **Move Planes to max Value** Set display plane to position of maximum field value
- **Copy** Add another display plane for this monitor
- **Delete** Remove monitor
- **Color Map** Choose different color scheme
- **Export** Save image as Stanford PLY, VTK ploygonal or STL facets
- **Export Animation** Save animation sequence as Stanford PLY, VTK ploygonal or STL facets in multiple files
- **Export Raw Data** Save field (Full or single Plane) in ASCII file as set in the 3D Plot Options

19.2. Waveguide Port Modes

This special field monitor is only available in the **3D Results** Tab if a waveguide source has been defined. Its coordinates and storage properties are defined in the waveguide source. It can only be displayed if the associated modes have been calculated (associated modes, e.g. `sub-1/wg_port_1.dbx` are available).

Hint: Waveguide Mode Display

Modes of waveguides are determined by solving a 2D eigenvalue problem of the cross sections right before the 3D simulation starts. In order to display waveguide modes in advance, the simulation should be run with a few time steps until mode field dumps e.g. `sub-1/wg_port_1.dbx` are available.

If higher modes are available at a port their field distribution can be viewed with a right click on the respective port, e.g. **GANLIB 1** and hovering the cursor over the available modes, see Fig19.5

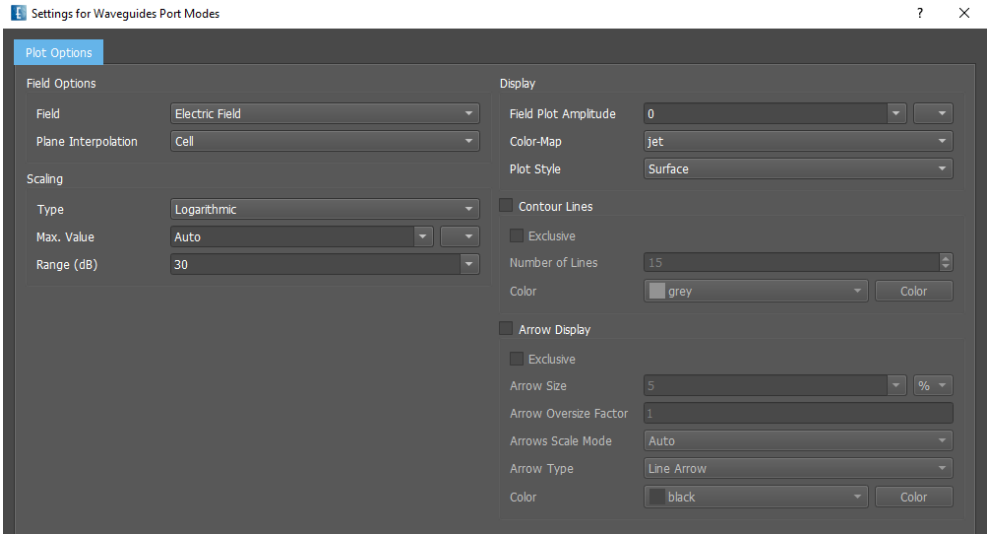


Figure 19.4.: Waveguide Port Mode Monitor

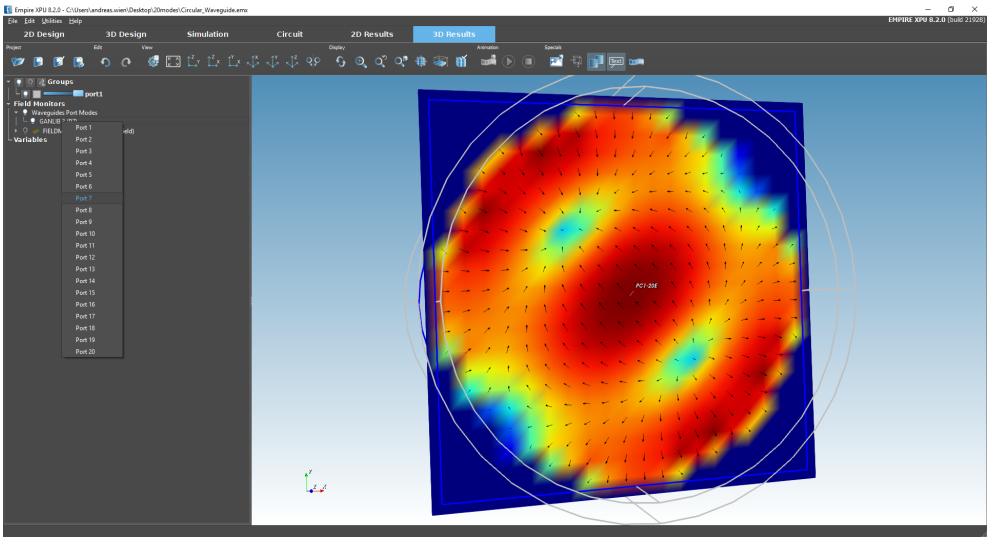


Figure 19.5.: Mode Display

Field Options

- **Field**: (In case of Planar field dump only E and H fields can be displayed)
  - Electric Field **E** in V/m
  - Magnetic Field **H** in A/m
  - Magnetic Flux Density **B** - calculated from $\mu_0 H$ in Vs/m^2
- **Plane Interpolation** Specifies the averaging in cells or nodes

Scaling

- **Type** Linear or logarithmic scaling
- **Max Value** If set to automatic, the maximum found in field dump is used
- **Range (dB)** Dynamic range. Effects on color map, legend and equi-lines

Display

- **Field plot amplitude** Hill plot display with maxmimum height determined by this value.
- **Color-Map** List of predefined color gradients
- **Plot style** Surface, wireframes or points

☐ Contour lines

if enabled lines of constant field strength are displayed

☐ **Exclusive** If enabled, only contour lines are displayed

- **Number of lines** Number of contour lines (depending on values defined in Scaling)
- **Color** Color of lines from list or custom color. **Field** applies line colors according to the field strength.

☐ Arrow Display

if enabled arrows representing direction and strength of the field are displayed

☐ **Exclusive** Vector display only

- **Arrow Size** Length of maximum value arrow in units
- **Arrow oversize Factor** Vector magnification
- **Arrows Scale Mode** If enabled vector length depends on field strength
- **Arrow Type** Cone, arrow, line
- **Color** Coloring according to field strength or single color

19.3. EM Far Field

With **EM Far field** definition near fields are recorded on a box surface (Huygens Surface) defined by the boundaries or specified by the user. During post processing the far field is obtained by a near-field-to-far-field transformation which requires that outside this surface is homogeneous space. As soon as this monitor is used, additional space is added around the structure automatically by the meshing algorithm. If the structure is too close to or even penetrating the Huygens surface this can lead to unphysical results and warning messages.

- **EM Far Field - Standard** Use boundaries as Huygens Surface
- **EM Far Field - Region** Specify Huygens Surface with a box
- **EM Far Field - External** Display monitor for external far field data

The far field pattern can be visualized in the **3D Results** Tab together with the structure or plotted in the **2D Results** Tab.

The **EM Far Field** monitor contains options for the storage, processing and display. In addition a point can be specified which acts as the center for display in the 3D Results.

Hint

Far field results are stored as electric field quantitites. This has to be considered when displaying directivity or gain in linear style. In order to obtain the power based value it has to be squared.

Hint: Application Note

More information is available in Chapter 29.2.7

19.3.1. Storage Options

General Settings

- **Frequencies for Storage** Here, single frequencies can be added or a set of equidistant frequencies can be entered.

Advanced Settings

- **Near-Field Filename** Is set automatically by default. For compatibility reasons or for special purposes it can be specified by the user.
- **Near field Sub-Sampling (per min. wavelength)** If set, a different grid than the FDTD mesh is used for storage of the near field.
- **Near field Sub-Sampling cell tolerance** Inserts a sub grid line if the FDTD cell is x% larger than the sample value

☐ **Broadband recording** Enables time-domain recording of the near field (high memory consumption) for a flexible post processing of the far field

☐ **Sim Cache Optimisation** Simulation speed improvement for far field recordings on machines with large number of CPU cores

☐ **Simulation TD Sub-Sampling (NB)** Speed optimization of field recordings in case of narrow band excitation

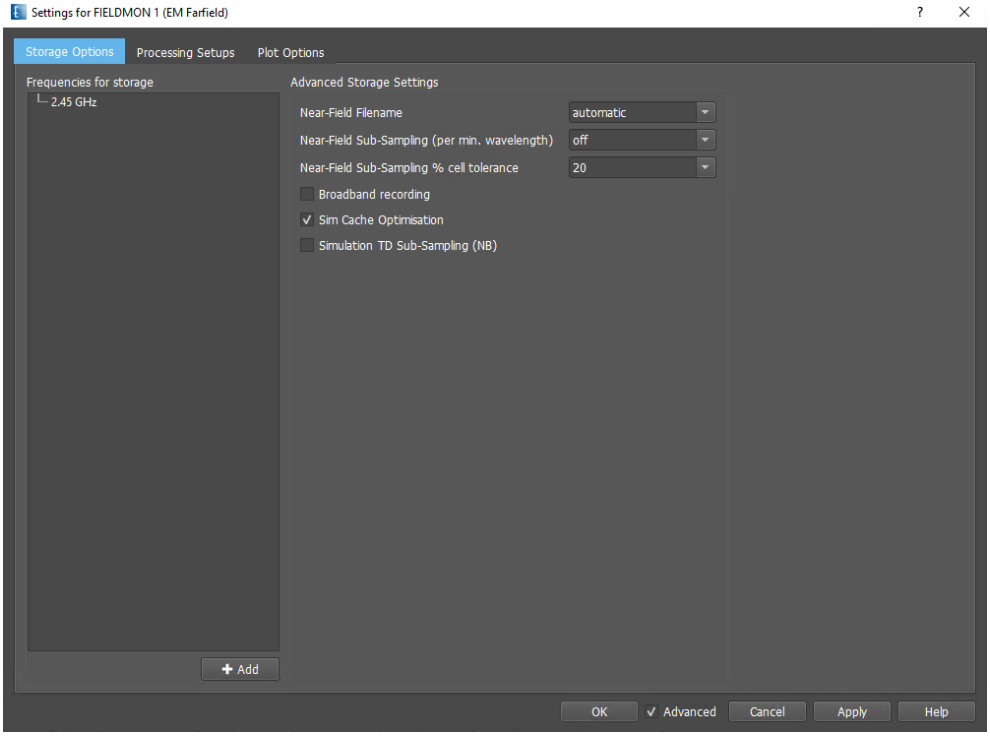


Figure 19.6.: Far-field storage definition

Hint: Farfield box size

The Farfield recording requires aprrox. $NF \times (NxNy + NyNz + NzNx) \times 96Bytes$ of memory where NF is the number of frequency points and Nx, Ny, Nz are the number of cells enclosed by this box. In order to save memory it is advantageous to keep the number of frequency points as low as possible.

19.3.2. Processing Setup

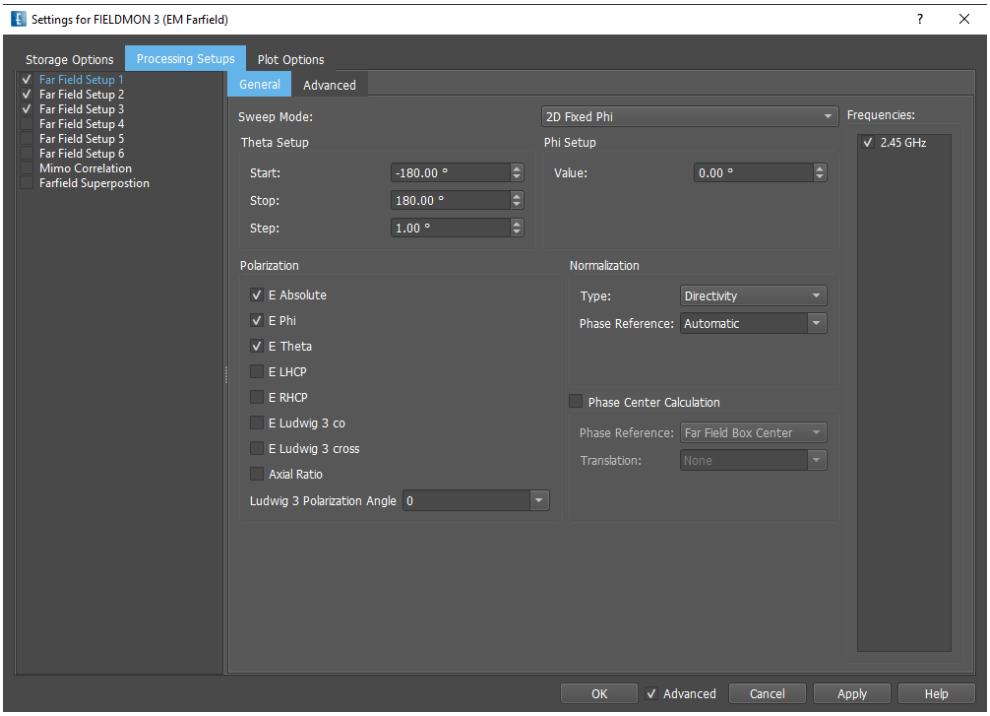


Figure 19.7.: Far field calculation

By default 3 far field set ups (FF Setup 1 ... FF Setup 3) are predefined and enabled which represent the far field pattern in the xz ($\varphi = 0$), yz plane ($\varphi = 90$) and a 3D pattern (Sweep Mode=3D). Another 3 far field set ups (FF Setup 4 ... FF Setup 6) are disabled by default and can be used if more far fields patterns should be calculated. Their defaults can be changed as described below, e.g. to refine the angle step or to change the normalizations.

General

- **Farfield Setup 1-6** If checked, this calculation is executed for each postprocessing. Can be disabled to save time in subsequent postprocessings
- **Sweep Mode**
  - **2D Fixed Theta** Single angle sweep, used for xy pattern if $\theta = 90$
  - **2D Fixed Phi** Single angle sweep, used for xz $\phi = 0$ or yz $\phi = 90$ patterns
  - **3D Sweep Phi and Theta** Sweep over both angles for obtaining a 3D pattern
  - **2D Multiple Theta Cuts** Single angle sweep, multiple files for Theta
  - **2D Multiple Phi Cuts** Single angle sweep, multiple files for Phi
  - **Backscattering** Use the opposite angle of the incoming plane wave, used for RCS calculations
  - **1D Point (frequency dependent)** Single angle transformation for different frequencies
  - **1D Sweep Phi and Theta (frequency dependent)** Multiple files for Phi and Theta

- **Angle Setup** Range values can be modified according to the selected **Sweep Mode**
- **Polarization** - Select the desired Farfield components
  - **E Absolute** - Magnitude of both polarizations
  - **E Phi component** - Azimuthal (horizontal) component of radiation pattern
  - **E Theta component** - Elevation (vertical) component of radiation pattern
  - **E LHCP** - Left-handed elliptical polarization component
  - **E RHCP** - Right-handed elliptical polarization component
  - **E Ludwig 3 co** - defined as $E_\theta cos(\varphi) - E_\varphi sin(\varphi)$
  - **E Ludwig 3 cross** - defined as $E_\theta sin(\varphi) + E_\varphi cos(\varphi)$
  - **Axial Ratio** - defined here as $|\frac{|eright|+|eleft|}{|eright|-|eleft|}|$
  - **Ludwig 3 Polarization Angle** Direction of the electric field (x: $\varphi = 0$, y: $\varphi = 90$). Far field result file name endings contain the angle if different to 0, e.g. el3co90.
- **Normalization**
  - **Directivity** Gives the pattern relative to an isotropic radiator. Independent from input power and insertion loss. The phase of the Directivtiy can be referred to a port phase if several ports have been excited.
    - **Phase Reference** for Directivity
  - **Gain** Directivity including ohmic losses. $G = DP_{rad}/P_{in}$ (radiated over accepted power)
    - **Port Reference** Power of specified port(s)
  - **Gain_Real** Realized Gain: Directivty including ohmic and reflection losses $G = DP_{rad}/P_{inc}$ (radiated over available power)
    - **Port Reference** Power of specified port(s)
  - **Excitation** Calculates the electric field strength at certain distance
    - **Excitation Type** Reference to source
      - **ef** e.g. plane wave excitation
      - **if1** e.g. current of port 1
      - **uf1** e.g. voltage of port 1
      - **p1** e.g. power of port 1
    - **Excitation Distance** Distance in m, if not specified unit sphere of 1m is used
    - **Excitation Weight** Source strength
  - **Maximum** Maximum normalized to 1 (0 dB)
- **Phase Center Calculation** If eneabled result is written in log window for each transformation
  - **Phase reference** - Phase of farfield relative to different origins
  - **Translation** - in m, e.g. $z\ 1e-3$ for 1 mm. Directions are separated by a comma.
- **Frequencies** - List of frequencies defined in the far field storage setup

The phase center is calculated internally independent of the E_ϕ, E_θ results and independent of the angle resolution and uses a cartesian grid. E.g. for calculation of the phase-center x-coordinate, two circles are used. Those two circles are in the yz-plane, the circle centers are slightly out of origin in the x-direction. An average discrete derivative of the Far Field between the two circles is calculated, which is directly linked to the x-coordinate of the phase center.

Hint

All far field normalizations are based on field strength. If plotted in linear style values have to be squared to obtain the power based quantities.

Hint: Efficiency

The efficiency is automatically calculated when the normalization is set to **Gain** or **Gain_Real** and derived from the port input power and radiated power.

Advanced

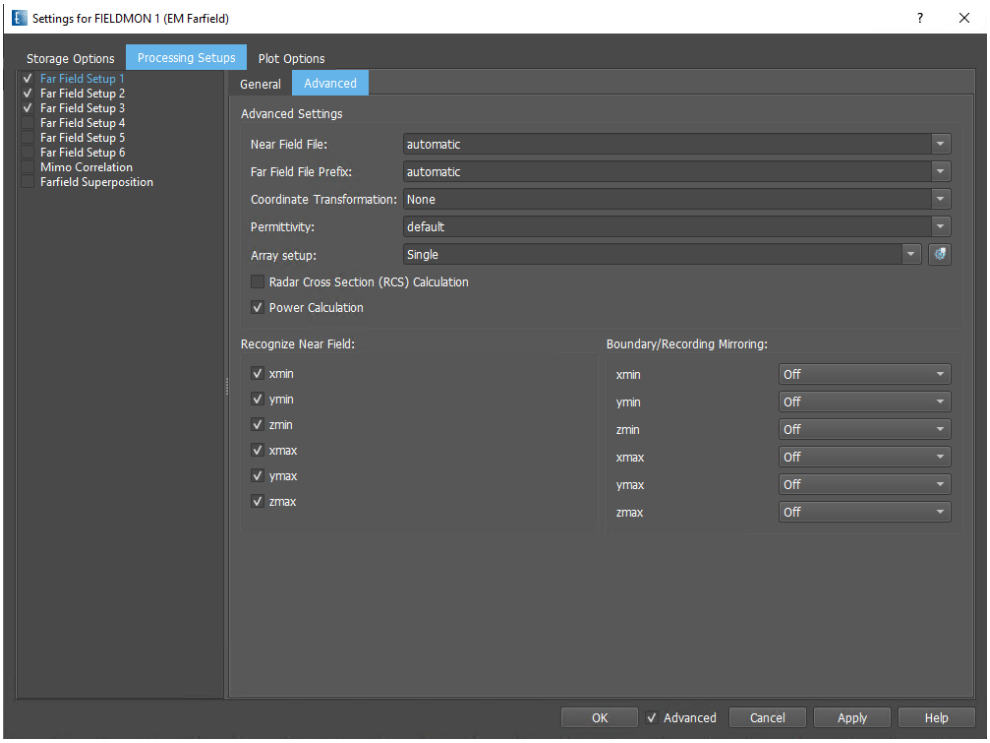


Figure 19.8.: Far field advanced processing setup

- **Near Field File** Name of recording defined in Storage options
- **Far Field File Prefix** - can be used to prevent overwriting of results, e.g. in order to compare Gain vs. Directivity
- **Coordinate Transformation** - Define a rotation of the far field in order to match antenna orientation to spherical coordinates of main axes.

  - **Main Axes, rectangular** - swap any main axis to z-axis
  - **Any 2 Angles** - ζ refers to θ and angle ψ refers to the angle ϕ in spherical coordinates
  - **Any Axis, any angle** - axis defined by x,y,z direction and rotation angle α. (Left hand rule: thumb directed in axis direction, fingers define rotation direction)
- **Permittivity** - Defines relative dielectric constant of space in far field. If set to default the permittivity of the **Background material** defined in the **Simulation Setup** is assumed.
- **Array Setup** - linear or rotational array obtained by far field multiplication. A string or table defines the shift, amplitude and phase for each element. This can be used to obtain the far field pattern for different amplitude and phase combinations.
  - **Edit Array Setup** Editor for setting below values
  - **amp** Amplitude of the feed
  - **phase** Phase of the feed
  - **dx, dy, dz** position of the element relative to the 0 element in **meter**
  - **dphi** angular position of the element relative to the 0 element in degrees
- **RCS calculation** - (Bi-static) Scattering cross section calculation
- **Power calculation** - determined from Poynting vector integration on Huygens box
- **Recognize Near Field** - Planes can be disabled if radiation in the respective direction should be neglected
- **Boundary/Recording Mirroring**
  - **Off** Mirror disabled
  - **Boundary** - Enable mirror for a half space, e.g. ground (adds 3 dB in far field calculation)
  - **Boundary (Symmetry)** - Enable mirror for symmetry
  - **Recording Box** - Enable mirror for a half space on a user-defined Hugens box, e.g. ground (adds 3 dB in far field calculation)
  - **Recording Box (Symmetry)** - Enable mirror for symmetry on a user-defined Hugens box

19.3.3. Mimo Correlation Setup

If more than 1 port is excited and the "Simulation Mode" is set to "Sequential Excitation" in the "Simulation Setup" the Envelope Correlation Coefficients can be calculated if enabled here. The coefficients will be stored in the **mimo** sub folder.

- Envelope Correlation Coefficient (ECC, based on Farfield)

  The calculation is based on the 3D Far Field patterns which have to be available
  - **MIMO Correlation** If checkmarked $k-1$ coefficents will be calculated if k is the number of ports
  - **Power distribution** isotropic or anisotropic channel distribution

    The anisotropic distribution is available for uniform spread in azimuth and Gaussian spread in elevation. If **gaussian_for_elevation** is set the **Gaussian Distribution Parameters** can be set as defined below.

The envelope correlation coefficient for port i and j is defined as

$$\rho_e^{ECC} = \frac{|\oint X_{PR} E_{\theta i} E^*_{\theta j} P_\theta + E_{\phi i} E^*_{\phi j} P_\phi d\Omega|^2}{\oint X_{PR} E_{\theta i} E^*_{\theta i} P_\theta + E_{\phi i} E^*_{\phi i} P_\phi d\Omega \oint X_{PR} E_{\theta j} E^*_{\theta j} P_\theta + E_{\phi j} E^*_{\phi j} P_\phi d\Omega}$$

with uniform spread in azimuth and gaussian spread in elevation:

$$P_\theta = e^{-(\frac{\theta(\pi/2-m_v)}{\sigma_v})^2} \quad P_\phi = e^{-(\frac{\theta(\pi/2-m_h)}{\sigma_h})^2}$$

Here E_θ, E_ϕ are the electric far field distributions, X_{PR} is the polarzation ratio, $m_{v(h)}$ is the mean angle of arrival and $\sigma_{v(h)}$ is the angular spread.

- Envelope of Diagram Correlation Factor (EDCF, based on S-Parameters)
  - **Enable EDCF caclulation** Calculate the envelope of diagram correlation factor

The envelope diagram correlation factor for 2 ports i and j is defined as

$$\rho_e^{DCF} = \frac{|S^*_{i,i} S_{i,j} + S^*_{j,i} S_{j,j}|^2}{(1-|S_{i,i}|^2-|S_{j,i}|^2)(1-|S_{i,j}|^2-|S_{j,j}|^2)}$$

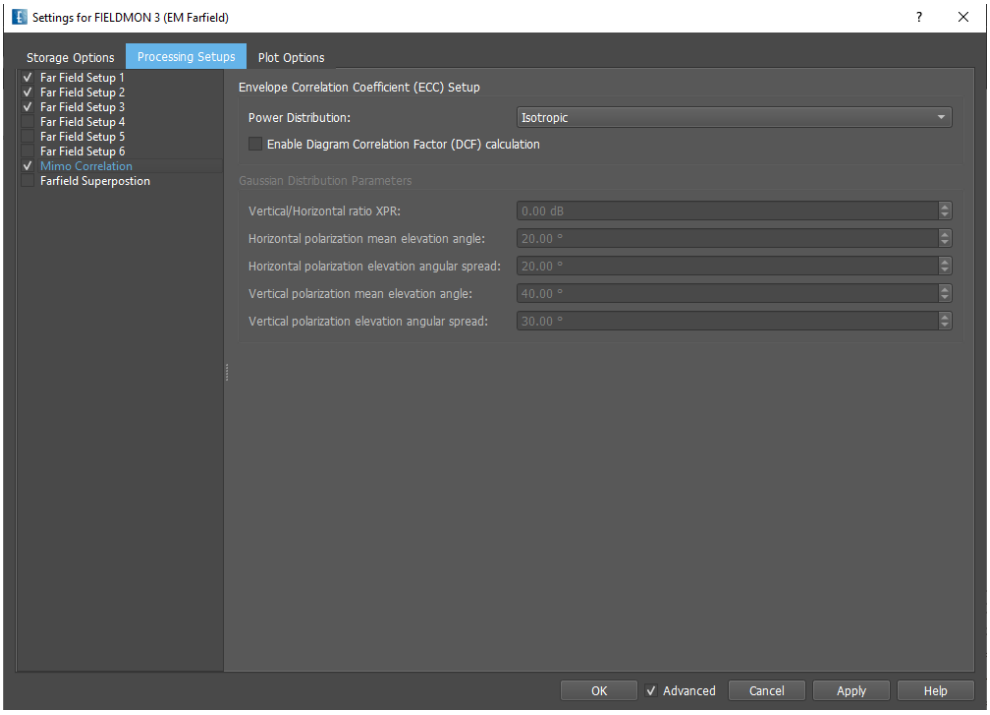


Figure 19.9.: Mimo Correlation Setup

19.3.4. Farfield Superposition

If more than 1 port is excited and the "Simulation Mode" is set to "Sequential Excitation" in the "Simulation Setup" the superposition can be calculated if enabled here. The coefficients can be entered as complex number, e.g. $1 + 0.5j$

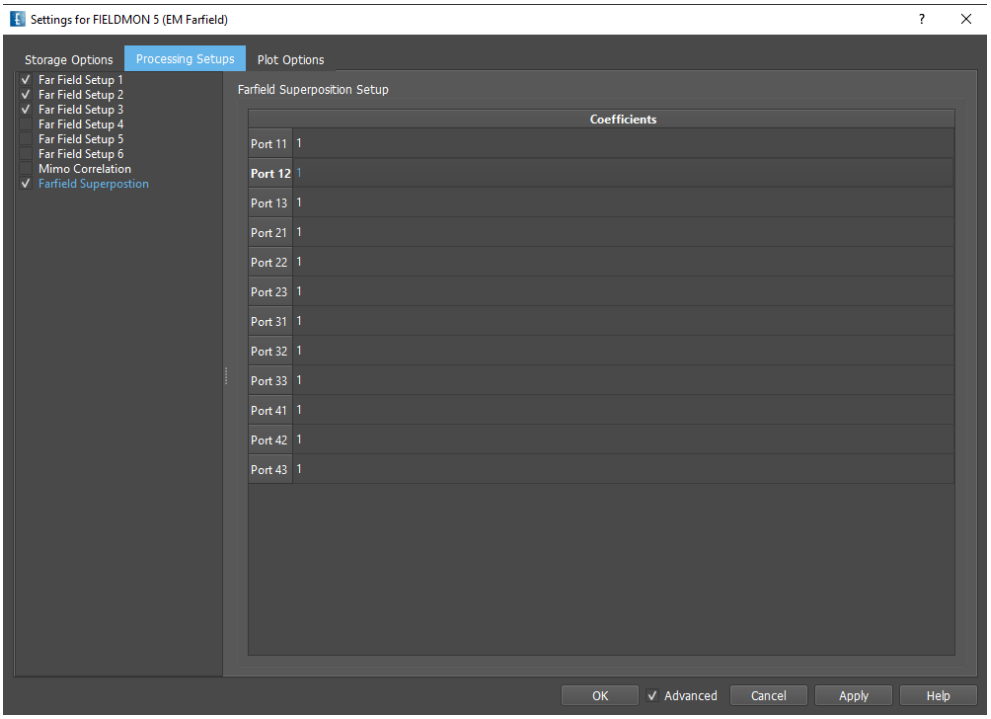


Figure 19.10.: Farfield Superposition Setup

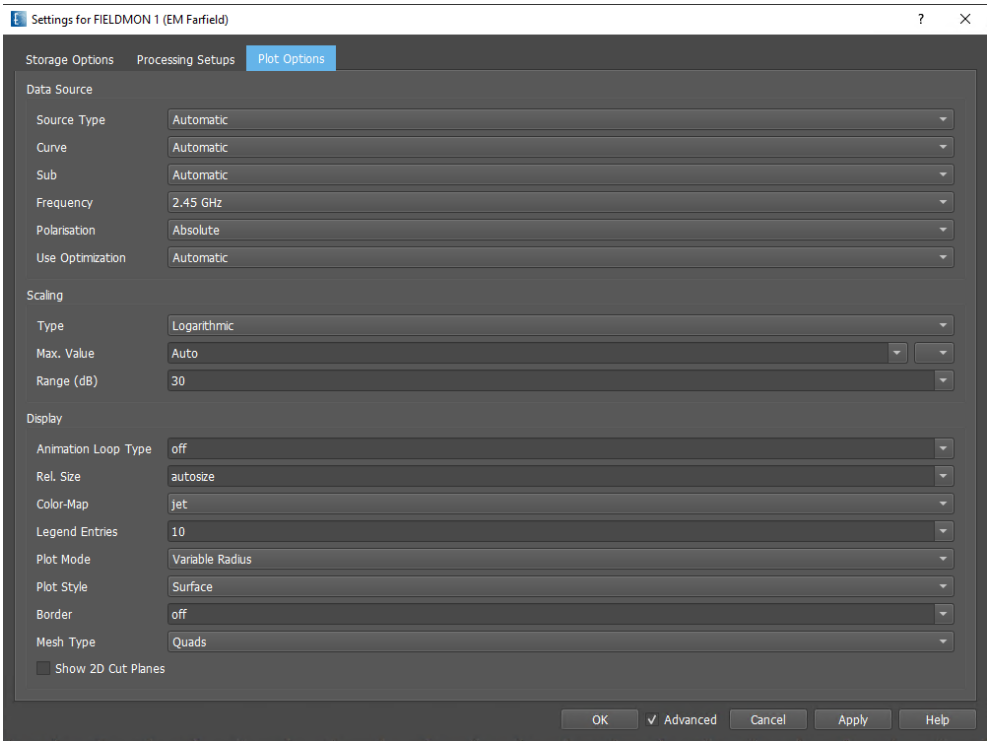


Figure 19.11.: Farfield Plot Options

19.3.5. Plot Options

In the **3D Results** Tab the far field pattern can be displayed relative to the structure. The center of the pattern display is defined by the **Display Origin** point.

- **Data Source**
  - **Source Type** - If set to Manual the result file can be chosen manually
  - **Curve** List of available far field results
  - **Sub** If multiple ports are excited the respective number can be selected
  - **Frequency** Select far field frequency for display
  - **Polarization** Select far field component for display
  - **Use Optimization** If variables have been swept the respective optimization folder can be selected
- **Scaling**
  - **Type** - Linear / logarithmic scaling
  - **Max. Value** - Can be limited to a certain value
  - **Range (dB)** - Maximum to Minimum range
- **Display**
  - **Animation Loop Type** - Frequency or Phase loop can be set
  - **Rel. Size** - Size of the maximum pattern elongation relative to drawing
  - **Color-Map** - Choice of different color gradients
  - **Legend Entries** - Set number of keys for field strength vs. colors
  - **Plot Mode** By default (var_radius) the pattern elongation is varied over the angle. It can also be displayed on a sphere with constant radius.
  - **Plot Style** - Display styles **fill, lines, points**
  - **Borders** - On/Off and line color for lines of constant angle
  - **Mesh Type** - Display style if lines are used as fill mode
  - ☐ **Show 2D Cut Planes** - Display cut planes as defined in **Processing Setups**

Hint: Application Note

More information is available in Chapter 29.2.8

19.3.6. 3D Result Options

The following options are available in the 3D Results Tab with a right click on the far field display

Add Marker

- Right click on field display adds a marker at cursor position. Multiple markers can be set
- Left click on marker and drag shifts the position of the marker
- The markers including position are listed in the field monitor
- The marker entries can be removed or moved to maximum in the list
- Marker font size, color and **Show Position** option can be set in the **3D Result Options**

FIELDMON

- **Edit** Open the **Plot Options** of this monitor
- **Hide/Show** Disable display of this monitor
- **Transparency** Set opaque value
- **Copy** Add another display plane for this monitor
- **Delete** Remove monitor
- **Color Map** Choose different color scheme
- **Scale** Choose different sizes (10 ... 100%) for display
- **Polarization** Choose different polarizations if available
- **Export** Save display as Stanford PLY, VTK ploygonal or STL facets

19.4. EM Probes

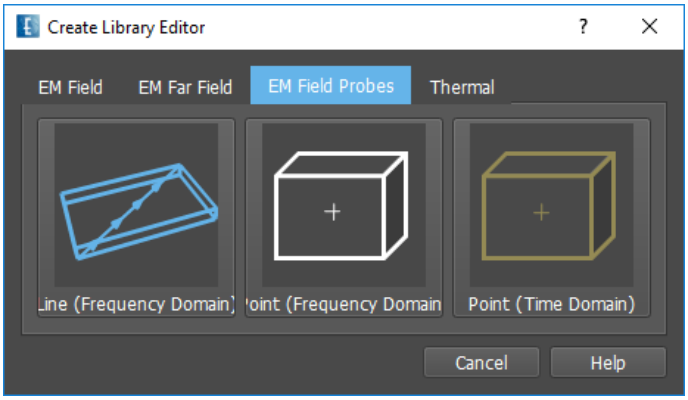


Figure 19.12.: Field Monitor EM Probes

With **EM Probes** near fields are recorded in points or along paths and can be evaluated with the plotting engine in the **Result** tab.

Hint

Field probes outside of the simulation domain are not supported. The field probe polarization is set in the field probe parameters

- **EM Field Line** Specifies a path by 2 points (Point 0 and Point 1) for field storage in frequency domain. For averaging purposes a volume is used for recording which is at least 2 cells larger in each direction.
- **EM Field Probe** Defines a point for field storage in frequency domain. For averaging purposes a volume is used for recording which is at least 2 cells larger in each direction
- **EM Time Domain Probe** Defines a point for field storage in time domain

Each **EM Probe** contains **Parameters** for the setup of storage and calculation.

19.4.1. Field Line

Storage Options

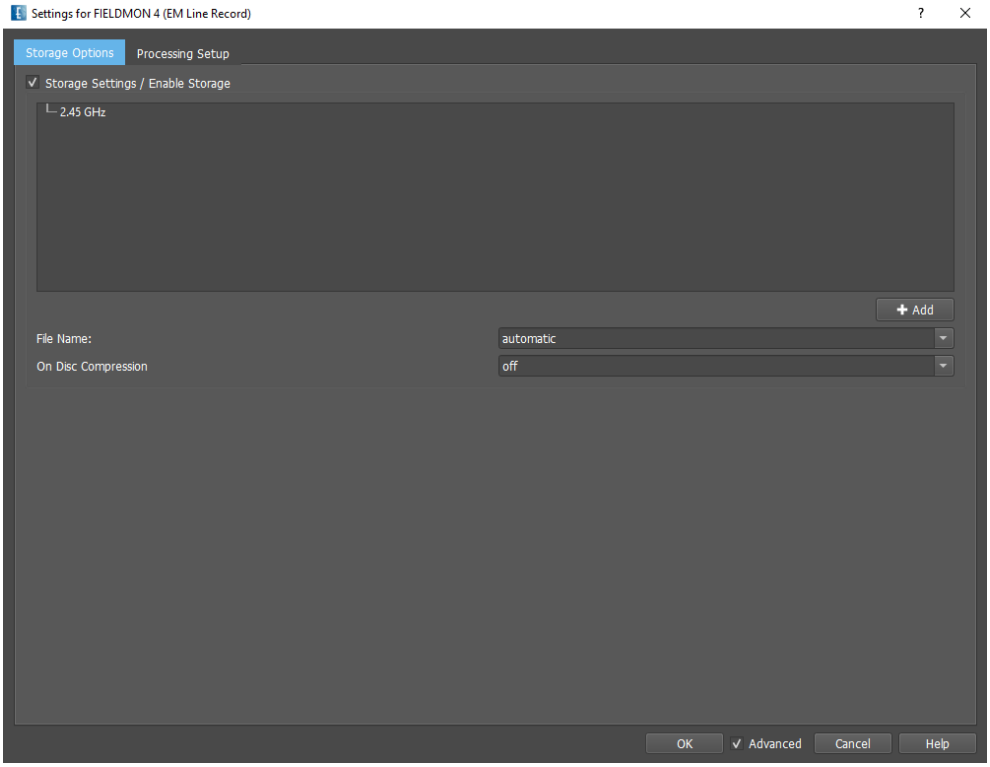


Figure 19.13.: Field line definition

Field lines are possible for frequency domain, only. Frequency dumps are generated by an on-line DFT during the simulation run, therefore the performance can slow down if many frequency points are used.

- **Storage Settings / Enable Storage** If disabled, no field dump storage, for additional calculation only
- **Storage Frequencies** To specify frequencies it is possible to define equidistant sequences or single points.
- **File Name** Is set automatically by default, but can be specified for compatibility reasons or special purposes

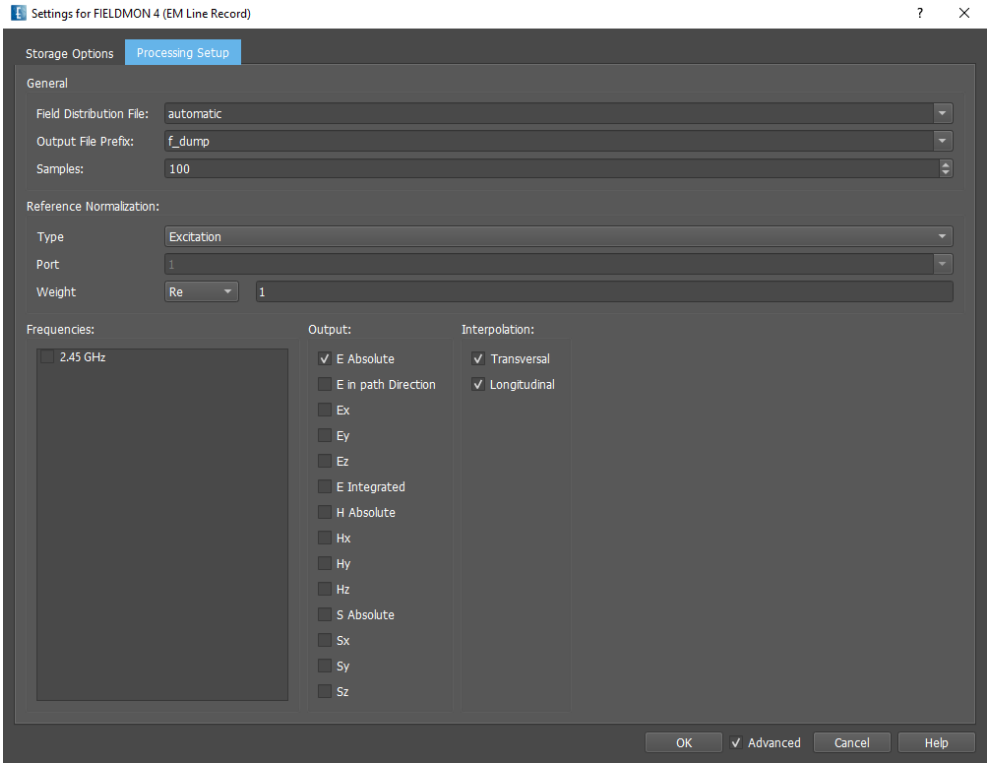


Figure 19.14.: Field line calculation

Processing Setup

This setup defines which components shall be stored and what interpolation scheme should be applied.

- **Field Distribution File** Name of the field dump defined in storage setup
- **Output File Prefix** Can be selected to distinguish different result files
- **Samples** Number of supporting points used in field path
- **Reference Normalization**
  - **Type** Current, Voltage or Power normalization
  - **Port** Number of reference port
  - **Weight** Normalization factor

Selections

- **Frequencies** - Frequency points which have been defined in field dump definition
- **Output** - Select component to be stored (electric / magnetic field, poynting vector)
- **Interpolation** - Select or deselect interpolation schemes

19.4.2. Point Field Probe (Frequency Domain)

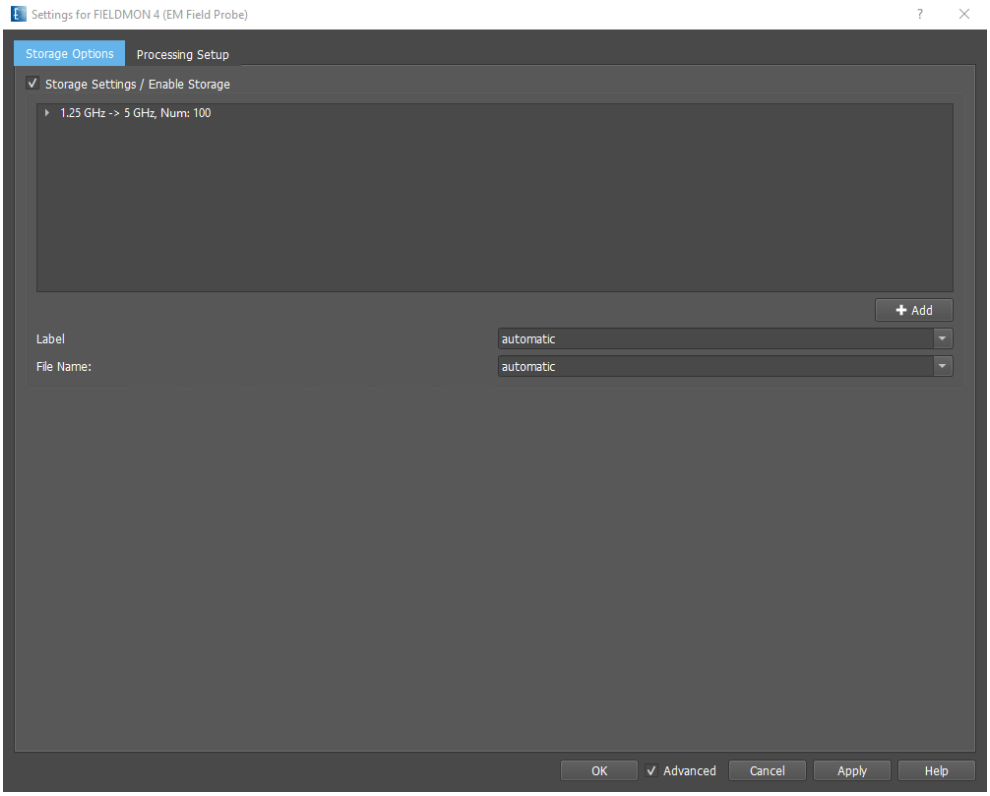


Figure 19.15.: Field probe definition

Storage Options

These field probes are possible for frequency domain, only. Frequency dumps are generated by an on-line DFT during the simulation run, therefore the performance can slow down if many frequency points are used.

- **Storage Settings / Enable Storage** If disabled, no field dump storage, for additional calculation only
- **Storage Frequencies** To specify frequencies it is possible to define equidistant sequences or single points.
- **Number** numeric value of probe as in result file name appearance, e.g. **emprobe_4.dbx**
- **Label** Name of field probe as appearance in 2D/3D Design display
- **File Name** Is set automatically by default, but can be specified for compatibility reasons or special purposes

Processing Setup

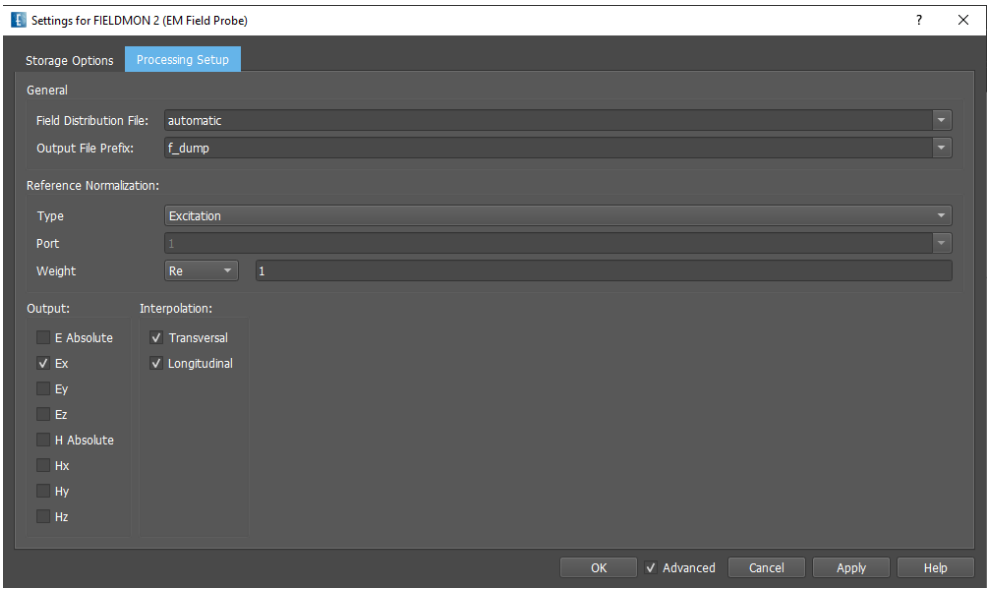


Figure 19.16.: Field probe processing setup

This setup defines which components shall be stored and what interpolation scheme should be applied.

- **Field Distribution File** Name of the field dump defined in storage setup
- **Output File Prefix** Can be selected to distinguish different result files
- **Reference Normalization**
  - **Type** Current, Voltage or Power normalization
  - **Port** Number of reference port
  - **Weight** Normalization factor
- **Output** - Select component
- **Interpolation** - Select or deselect interpolation schemes

19.4.3. Point Field Probe (Time Domain)

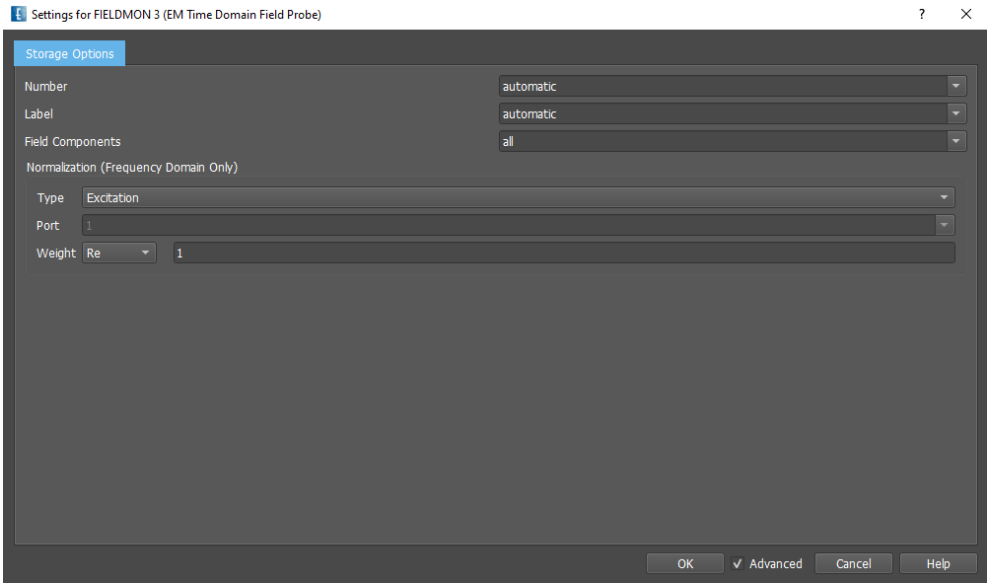


Figure 19.17.: Time Domain Field probe definition

This field probe records electric or magnetic field components in time domain of the cell which is closest to the specified Point 0. No interpolation is used and the cell component location is according to the Yee cell convention. In post processing the DFT will be applied for the associated result in frequency domain.

- **Number** of probe to differentiate different probes. Is set automatically by default, but can be specified by the user
- **Label** Name of field probe as appearance in 2D/3D Design display
- **Field Components** By default all six components are used but may be limited to single components by entering, e.g. **Ez**.
- **Normalization (Frequency Domain Only)**
  - **Type** Current, Voltage or Power normalization
  - **Port** Number of reference port
  - **Weight** Normalization factor

19.5. Thermal Field Monitor

For thermal simulation a thermal display monitor and a thermal point probe can be defined.

19.5.1. Thermal Display

A thermal monitor saves the temperature distribution for the complete volume. After simulation the display plane can be selected with the slider on the side menu. Additional planes may be added for display. The following parameters can be adjusted for display.

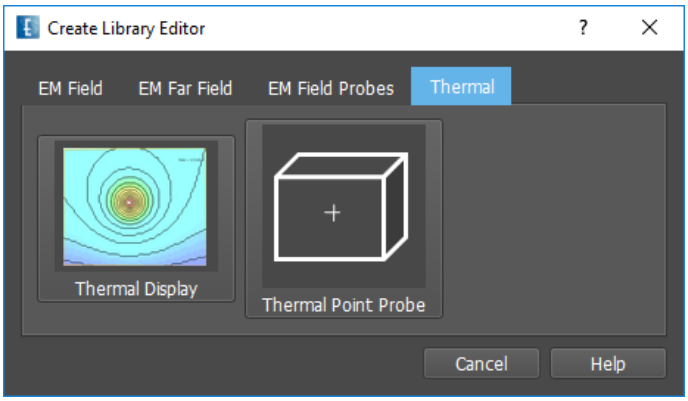


Figure 19.18.: Thermal Monitor

- **Data Source**
  - **Source Type** - If set to **Manual** the file name can be specified
  - **Sub** - Result folder selection if multile ports have been used
  - **Use Optimization** - Browse variation folders
- **Display**
  - **Display Mode**
    - **bottomside** - displays temperature distribution of the bottom surface
    - **maximum** - displays the maximum temperature of all planes
    - **minimum** - displays the minimum temperature of all planes
    - **plane** - displays the temperature of the selected plane
    - **topside** - displays temperature distribution of the top surface
  - **Minimum Value** - sets the minimum value for temperature display
  - **Maximum Value** - sets the maximum value for temperature display
  - **Field Plot Amplitude** - Set elongation of hill plots
  - **Color-Map** - Color mapping gradient
  - **Legend Entries** - defines the number of colors and legend entries used to display the temperature
  - **Legend Unit** - defines the unit (Kelvin, Celsius, Farenheit) of the temperature
- **Contour Lines** If enabled, lines of constant temperature are displayed
  - **Exclusive** If enabled, color map is suppressed
  - **Number of Lines** - defines the number of lines of constant temperature
  - **Color** - defines the color of contour lines

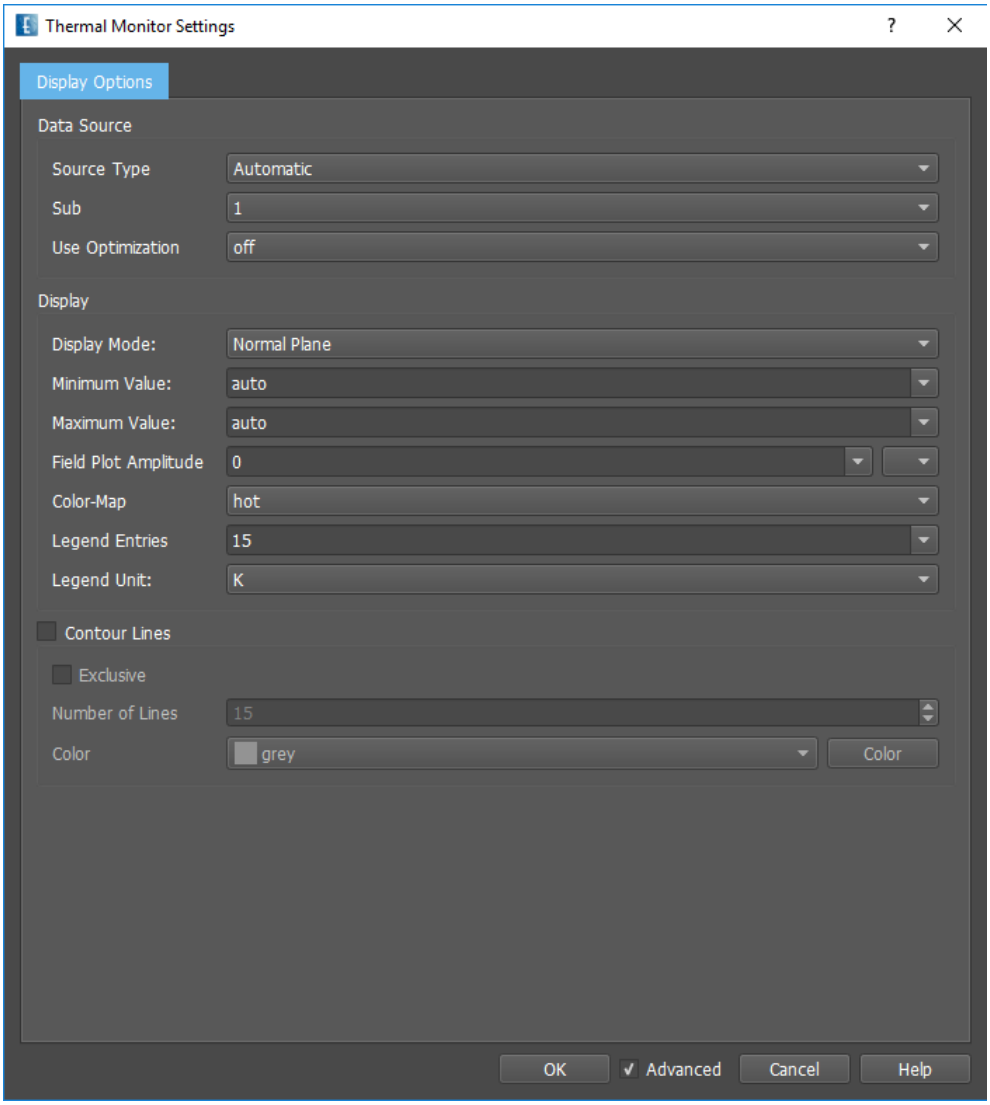


Figure 19.19.: Thermal Monitor Parameters

19.5.2. Thermal Point Probe

For the record of the temperature in single points this monitor is available. The record number may be adjusted in the thermal probe parameters. The location can be set in the navigation tree.

20. Sources

The following section describes the different sources which are available by pressing the **Create Source** button. The sources can be categorized into three types:

- **Ports** - Excitation (and termination) of waveguides, circuits or antennas
- **Volume Sources** - Excitation by a plane wave or near field source
- **Thermal Source** - Excitation for thermal simulation

Hint: Coordinate System of Sources

All sources have to be created in the World Coordinate System (WCS) and (with the exception of the **Wire Port**) must be oriented parallel to the x, y or z axes. A port creation in a Local Coordinate System (LCS) is not possible.

20.1. Ports

For all ports the following hints are valid:

Hint: Ports

- Ports record voltage and current time signals
- The direction of a port must coincide with one of the major axes. The exception is a **Wire Port**
- Ports are compound objects. Their basic elements (e.g. voltage, current, load, excitation boxes) can be displayed by selecting the port and pressing the icon **Advanced - Library Elements - Explode Library Element.** (Use **Undo** to recover the port.)

Hint: Port environment

Ports are library objects which consist of different elements, such as resistive objects and measurement boxes. In order to properly model loads and measure correct voltages or currents, the ports may not overlap with conductors to prevent full or partial short-circuit of the port elements.

The current flowing out of the port with number n is named itn in time domain and ifn in frequency domain. The voltage at port n is named utn and ufn in time and frequency domain, respectively. The results ifn and ufn can be used to normalize frequency domain results so that the structure is excited by a constant current source of 1A or a constant voltage source of 1V, respectively. After simulation the voltages and currents will be separated into reflected and incident wave quantities. Then a DFT is applied to the time signals in order to obtain the frequency domain results so impedance and scattering parameters can be derived. The impedance seen from port n is named as $Zn.in$.

Hint: Reference Impedance

The default reference impedance for separating incident and reflecting wave is 50Ω and is used for S-Parameter calculation. The load impedance is used for termination. If this value is different and should be used as reference impedance the switch **Use internal impedance as reference impedance** should be set to **Rcopy=On** in the port settings. In case of QTEM ports, this switch is set to **Rcopy=On** by default.
The actual value is printed in the simulation log window and in the **Simulation Setup - Port Setup - Port Configuration (Advanced)** of the the simulation tab.

With **Rcopy=Off** the current settings in the Port Setup remain unchanged and do not set back to 50Ω automatically.

Usage

To insert a port into the current workspace:

1. Click the button **Create Port** 
2. Select one of the available port types in the palette
   - In 3D Design
     - Click the points following the guide at the left bottom, or
     - Press **Tab** key to cancel cursor input and enter values in editor
   - In 2D Design
     - Enter the coordinates values inside the editor, or
     - Right click on a **Point** and use **Pick Coordinate** to apply the cursor for input
3. Adjust Port Parameters with **Edit Settings**
4. Press **Close**

The view can be adjusted during port creation in order to find the correct position of the port.

The group height can be used in advance for some port definitions which either represents

- Start and End height = bottom and top side of the signal strip (msl, triplate, coplanar),
- Start height = midpoint of the center conductor (circular coaxial, square coaxial),
- Start height = center of the circular waveguide or lower side of the rectangular waveguide,
- Start and End height = thickness of the port (in-plane port), or
- Start and End height = height of the port (perpendicular port)

After a port is created the **Object Editor** can be opened again using **Open/Edit Object**  or with double click on the respective item in Groups list. With **Edit settings** the port parameters can be acceessed which contain settings such as geometry, material, impedance, delay, port type, port number, etc.

Some properties of the port may also be defined parametrically, e.g. for variation and optimization purposes.

20.1.1. Port Setup Wizard

If many ports are defined the editing of their major parameters can be done easily with the aid of the **Port Setup Wizard** . Before starting the port setup wizard it is recommended to select a view so that all ports are visible and do not overlap.

Hint: Exciting Port Parameters

The following parameters only apply for exciting ports:

- Amplitude
- Phase Shift
- Time Delay
- Broadband Phase Shift

For non-exciting ports these values are disregarded

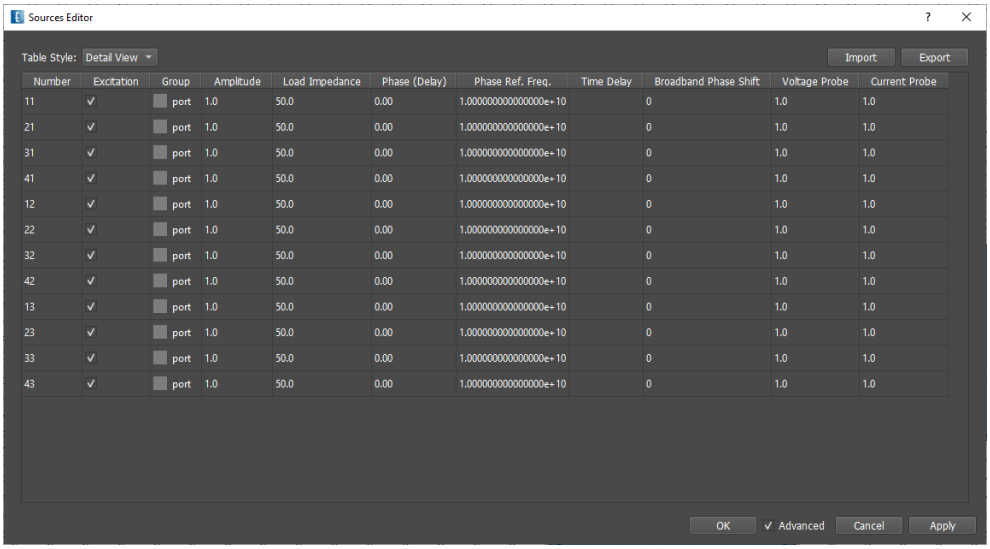


Figure 20.1.: Port Setup Wizard

- **Table Style**
  - **Detail View** The table is arranged with port parameters in columns. Sorting can be apllied by clicking on the respective column header.
  - **Array View** The port parameters are arranged in a matrix according to their positions in the current view, e.g. top view. A **Calculator** is available to apply position dependent equations.
- **Import** A port parameter setup can be imported which has been exported from another project
- **Export** Export current port parameters in a file **.dat**
- **Parameters**
  - **Number** Port numbering
  - **Excitation** Checkmarks for port excitation or termination only

- **Amplitude** Value in Ampere (current sources) or Volt (voltage sources)
- **Load Impedance** Port termination in Ohm, 0 and infinite are supported
- **Phase Shift** Angle in degrees valid for a single frequency
- **Phase Ref. Frequency** Reference frequency in Herz used to derive time delay for phase shift
- **Time Delay** Value in seconds
- **Broadband Phase Shift** Angle in degrees valid for the frequency range. (For AC simulations, only)
- **Voltage Probe** Weight factor for voltage recording
- **Current Probe** Weight factor for current recording

Hint: Broadband Phase Shift

If multiple ports are excited simultaneously a broadband phase shift may be defined. This is only possible for modulated pulses. This can be set in the **Simulation Setup - EM Setup** by checkmarking the **Modulated Pulse** switch. If the **Structure Type** is set to **User Setup** the modulation has to be defined by the user.

20.1.2. QTEM Ports

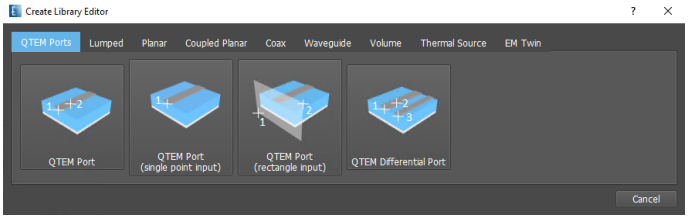


Figure 20.2.: QTEM port types

QTEM (Quasi-TEM) ports excite and terminate a general transmission line by solving an eigenvalue problem of the line cross section. There are two types:

- QTEM port (Signal-Ground system)
  - **dual point input** - Classic port definition with point 1 for signal line start and point 2 for wave direction. Cross section is derived from signal to ground spacing.
  - **single point input** - Simplified port definition with point 1 for signal line start. The wave direction is derived from input view and the cross section is derived from signal to ground spacing.
  - **rectangle input** - 2 points span a rectangle for the cross section. The wave direction is derived from input view. The signal line is determined as the smallest electrode inside the cross section. If another signal line should be used another input method has to be chosen.
- QTEM differential port (Signal-Ground-Signal system)
  - **triple point input** - Port definition with point 1 for signal line start, point 2 for wave direction and point 3 for second signal line start. Cross section is derived from signal to ground spacing.

The ground electrode is detected automatically inside the cross section perpendicular to the line within the area defined by the **Port Size**. After creation the signal lines are marked with solid green lines, the ground is marked with blue solid lines inside the cross section.

Usage

- It is assumed that the transmission line is already defined and the QTEM port should be applied in a plane perpendicular to the line direction
- **3D** Adjust the view so the signal line can be picked by the cursor. Enter points according to the selected QTEM type
- **Object Editor** Adjust points as depicted in the image, optionally edit name and group
- **Library Editor** Port Parameters can be set with **Edit Settings**

Calculation

- The line cross section is analyzed within a certain area as displayed with a large rectangle. Its size can be set with the parameter **Port Size** in the **Library Editor**
- The cross section should be large enough to accurately calculate the QTEM mode but small enough not to disturb radiation or neighbouring ports.
- The dominant mode is solved for the frequency defined in **Additional Parameters** and the electric field is calculated in the cross section and used for excitation
- Impedance and effective permittivity are calculated and can be evaluated in the **Info** window
- The boundary conditions at the cross section borders are assumed either magnetic or electric for mode calculation

Cross Section

- Backward waves are terminated by an absorber with a size of the cross section
- Voltages and currents are recorded at port location
- The cross section may have arbitrary permittivity distribution. Losses are disregarded.
- Magnetic materials are not supported in the QTEM port cross section.
- Cross sections of neighboring QTEM ports should not overlap
- A perfect magnetic conductor (PMC) block is used shield regions before and after the port, so the port can be placed at any position of the line

Hint: QTEM absorber

The absorber size is as large as the cross section. Therefore QTEM ports in the vicinity of antennas may absorb part of the radiation.

Hint: Multiple ground electrodes

In case of multiple (unconnected) ground electrodes the dynamic mode can be different to the static mode. The modal difference will be calculated and a limit can be set in **Advanced Parameters** in case the static mode should be used. It can be advisable to use ground connections in the port plane.

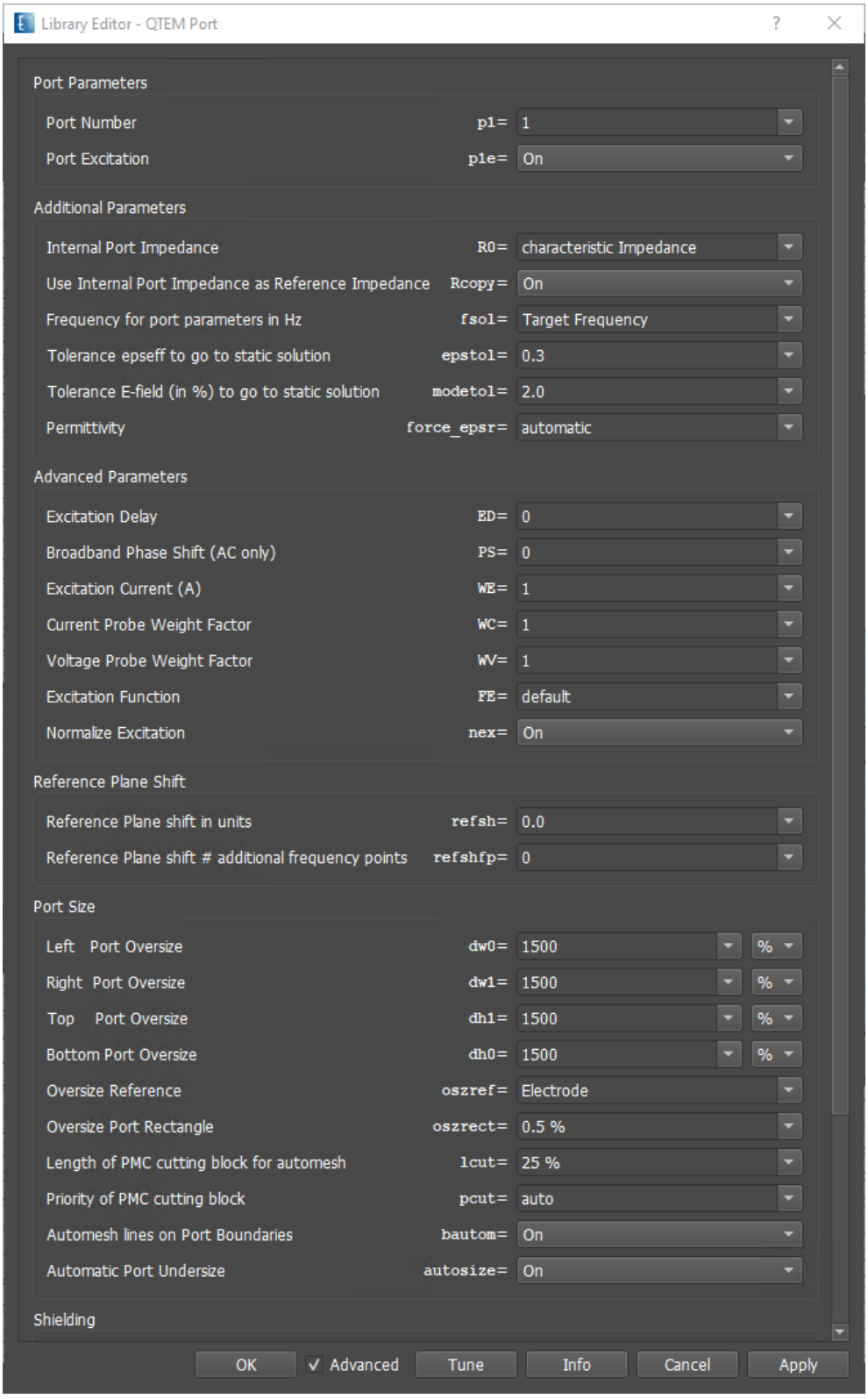


Figure 20.3.: Parameters of the QTEM Port

QTEM Port

The input method is to define 2 points

1. Active electrode start point (e.g. click on strip edge)
2. Active electrode wave direction (e.g. click on strip apart from edge)

After creation, the ports can be edited with **Open/Edit Object** 

Object Editor

- **Name** Optional name for display in group
- **Group** Group used for port
- **Points** x,y,z define the port coordinates

After creation the following Parameters can be edited with **Edit Settings**

- **Port Parameters**
  - **Port Number** $p1 \geq 1$, to be adjusted for multiple ports
  - **Port Excitation** $p1e = On/Off$, **Off** - port only used for termination
- **Additional Parameters**
  - **Internal Port Impedance** $R > 0$ in Ω, internal load of source. If set to **characteritic Impedance** the value will be calculated automatically
  - **Use internal impedance...** $Rcopy = On/Off$. If set to **On** this value will be used for S-Parameters, see **Hint: Reference Impedance** in 20.1
  - **Frequency for port parameters in Hz** To be changed if different from **Target Frequency** defined in the **Simulation Setup**
  - **Tolerance epseff...** The static eigenmode is used as excitation if the effective permittivity difference of the dynamic mode is larger than this value
  - **Tolerance E-field ...** The static eigenmode is used as excitation if the electric field deviation of the dynamic eigenmode is larger than this value
  - **Permittivity** Manually force permittivity of cross section
- **Advanced Parameters**
  - **Excitation Delay** $ED \geq 0$ in s or as abbreviation `angle@frequency`, used for narrow band phase shift in multiple simultaneous excitations
  - **Broadband Phase Shift (AC only)** $PS \geq 0$ in $degree$, used for broadband phase shift in multiple simultaneous excitations (only possible for modulated pulses)
  - **Excitation Current** WE, Value of Current source in A
  - **Current Probe Weight Factor** WC, used for, e.g. magnetic symmetry plane
  - **Voltage Probe Weight Factor** WV, used for, e.g. electric symmetry plane
  - **Excitation Function** FE, used for multiple simultaneous excitations with different pulse shapes
  - **Normalize Excitation** nex, used to normalize current to 1A
- **Reference Plane Shift**
  - **Reference Plane shift in units** Distance from port start point

  - **Reference Plane shift # additional frequency points** Improvement for dispersive lines

- **Port Size** Determines the size of the line cross section to be analyzed. (100% refers to the distance between signal and ground). The cross section covers the EM field which is used for excitation and termination. A large cross section is more accurate in general. A small cross section can be advantageous if the port is in the near of an antenna to avoid absorption of the radiation. So the choice of cross section size is a compromise.
  - **Left Port Oversize** lower border of cross section in units or percent
  - **Right Port Oversize** upper border of cross section in units or percent
  - **Top Port Oversize** upper border of cross section in units or percent
  - **Bottom Port Oversize** lower border of cross section in units or percent
  - **Oversize Reference** Object type
  - **Oversize Port Rectangle** Oversize factor
  - **Length of PMC cutting block** Thickness of magnetic wall which is placed behind the load and excitation plane
  - **Priority of PMC cutting block** Geometric priority with respect to intersecting objects
  - **Automesh lines...** If enabled, mesh lines are placed at QTEM cross section borders
  - **Automatic Port Undersize** Automatic reduction of port size for eigenmode analysis
- **Shielding** For shielding purposes an absorber can be placed at the QTEM cross section borders
  - **Shielding length in units** length of the absorber
  - **Absorber on shield** On or off
  - **Shield priority** Geometric priority in case of intersection with other objects
- **Display**
  - **Drawing Mode** Full or fast display
  - **Info Mode** Determines the content in the Info Window
- **Buttons**
  - **Tune** Impedance and effective permittivity calculation of the port cross section using variables. The slider of the variables can be adjusted and the line values can be calculated and printed in the Tuning Window
  - **Info** Impedance and effective permittivity calculation of the port cross section

QTEM Differential Port

The input method is to define 3 points

1. Active electrode start point (e.g. click at strip edge)
2. Active electrode wave direction (e.g. click at strip apart from edge)
3. Opposite electrode (e.g. click at other strip edge)

After creation the following **Parameters** can be edited

- **Port Parameters**
  - **Differential Port Number** $p1 \geq 1$, to be adjusted for multiple ports
  - **Common Mode Port Number** $p1 \geq 1$, to be adjusted for multiple ports
  - **Differential Excitation** $p1e = On/Off$, **Off** - port only used for termination
  - **Common Mode Excitation** $p1e = On/Off$, **Off** - port only used for termination
- **Additional Parameters** as in QTEM Port

20.1.3. Lumped Ports

The library elements

- In-plane port
- Perpendicular port
- Wire port

are lumped ports which can be inserted between two conductors to

- record voltage between the two conductors
- record the current flowing through the conductors
- terminate the two conductors with a resistor (default 50Ω)
- excite a current with 1 A if excitation is switched to **On**

The difference between the in-plane port and the perpendicular port is the method of creation, only. The wire port defines a small lumped port with wire connections which may be off-axis.

Common Parameters:

1. **Advanced Parameters**
   - **Port Load Impedance**, $R > 0$ in Ω, internal load of current source
   - **Use Internal Port Impedance as Reference Impedance** $Rcopy = On/Off$. If set to **On** this value will be used for S-Parameters, see **Hint: Reference Impedance** in 20.1
   - **Excitation Delay** $ED \geq 0$ in s or as abbreviation `angle@frequency`, used for narrow band phase shift in multiple simultaneous excitations
   - **Broadband Phase Shift (AC only)** $PS \geq 0$ in $degree$, used for broadband phase shift in multiple simultaneous excitations (only possible for modulated pulses)
   - **Excitation Current**, WE, Value of Current source in A
   - **Current weight factor**, WC, used for, e.g. magnetic symmetry plane

- **Voltage weight factor**, WV, used for, e.g. electric symmetry plane
- **Excitation Function**, FE, used for multiple simultaneous excitations with different pulse shapes

2. **Automesh Parameters**
   - **Bounding Box oversize**, $d0 > 0$ additional space to boundary
   - **Bounding Box length oversize**, $d1 > 0$ additional space to boundary in length direction

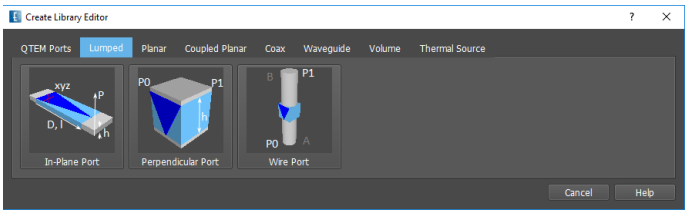


Figure 20.4.: Lumped port types

In-Plane Port

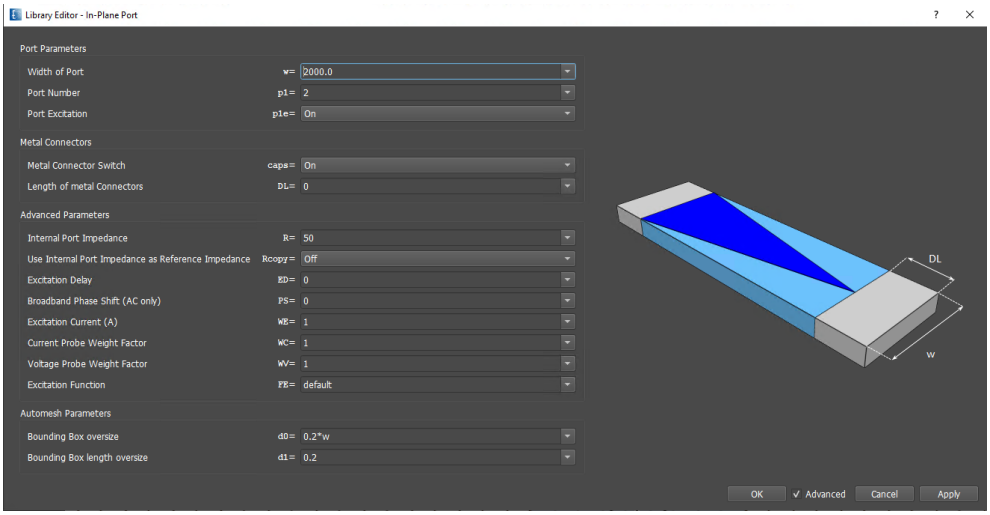


Figure 20.5.: Parameters of In-plane Port

Hint

This port type is best suited to connect 2 conductors on the same layer.

- Plane and Direction determine the orientation of the port
- x,y,z define the start coordinates
- l,h define the length and the height of the port
- The lumped port may not be placed at the boundary.

The input method is to define several points

1. Select plane or grid of the port plane
2. Start point of port
3. End point of port
4. Port height
5. Port width

After creation, the port can be edited with **Open/Edit Object**

Object Editor

- **Name** Optional name for display in group
- **Group** Group used for port
- **Plane** Port plane
- **Direction** Length and port direction
- **Base Point** Coordinates of start point

After creation the following Parameters can be edited with **Edit Settings**

1. **Port Parameters**
   - **Width of Port**, $w > 0$ in drawing unit
   - **Port Number**, $p1 \geq 1$, to be adjusted for multiple ports
   - **Port Excitation** $p1e = On/Off$, **Off** - only used for termination
   - **Length of metal connectors**, $DL \geq 0$, used if port connections have to be enlarged

Perpendicular Lumped Port

Hint

This port type is best suited to connect 2 conductors between different layers.

- The length of the port is defined by the height
- The cross section of the port is defined by the Points P1 and P2.
- The direction of the perpendicular lumped port is defined by the weight factors. Set excitation WE, voltage WV and current WC weight factors to -1 to invert the direction
- The lumped port may not be placed at the boundary.

The input method is to define several points

1. Select plane or grid of the cross section
2. Point 1 of cross section
3. Point 2 of cross section
4. Port length

After creation, the port can be edited with **Open/Edit Object**

Object Editor

- **Name** Optional name for display in group
- **Group** Group used for port
- **Height** Length and direction of port
- **Points** Coordinates of cross section

After creation the following Parameters can be edited with **Edit Settings**

1. **Port Parameters**
   - **Port Number**, $p1 \geq 1$, to be adjusted for multiple ports
   - **Port Excitation** $p1e = On/Off$, **Off** - only used for termination
   - **Length of metal connectors**, $DL \geq 0$, used if port connections have to be enlarged

Wire Port

Hint

This port type is best suited to connect 2 conductors between arbitrary 3D points.

The wire port consist of a small lumped port which is oriented in one of the major axes and connecting wires which may be oriented arbitrarily.

- The start (A) and end (B) of the port is defined by the Points P1 and P2.
- The lumped port may not be placed at the boundary.

The input method is to define several points

1. Point 1 for port start
2. Point 2 for port end

After creation, the port can be edited with **Open/Edit Object** 

Object Editor

- **Name** Optional name for display in group
- **Group** Group used for port
- **Points** Coordinates of port start and end

After creation the following Parameters can be edited with **Edit Settings**

1. **Port Parameters**
   - **Wire Diameter**, $d \geq 0$, size of the connecting wires
   - **Inner Port Length**, $l \geq 0$, size of the lumped port
   - **Port Number**, $p1 \geq 1$, to be adjusted for multiple ports
   - **Port Excitation** $p1e = On/Off$, **Off** - only used for termination
2. **Advanced Parameters** identical to in-plane port.
   - **Wire Resolution** specifies the arc resolution of the wire circular cross section

20.1.4. Feedline Ports

Feedline ports are dedicated to some popular transmission lines. They impress an electric field between signal and ground electrodes and require a certain port length in order to establish the desired (quasi-) TEM mode. They are available for compatibility reasons and may be substituted by the QTEM ports in most applications. In contrast to QTEM ports the terminating load is a sheet resistor between signal and ground lines which has less influence on radiation than the absorber of the QTEM cross section.

The feedline ports

- Planar
- Coupled Planar
- Coax

are either concentrated or absorbing ports which are used to feed a transmission line and

- record voltage between signal and ground conductors
- record the current flowing through the signal conductor
- in case of concentrate port terminate signal and ground conductor with a resistor (default 50Ω)
- excite a quasi-TEM mode with a current of 1 A if excitation is switched to **On**

These ports may be defined at the boundary or at any position inside the simulation domain.

Hint: Concentrated Ports

- A concentrated port may not be placed at the boundary.
- The ports contain a load (port impedance)
- If the port is excited, a current source with 1A is added as an excitation box to the port parallel to the load.

If the line impedance is not known and should be evaluated it is advisable to place the port at the boundary and terminate the line with an absorbing wall instead of a fixed port load.

Hint: Absorbing Ports

- An absorbing port must be placed at an absorbing boundary
- In case of excitation the XPU algorithm can not be applied (slow simulation)

For proper evaluation of concentrated port parameters, voltages and currents have to be evaluated sufficiently apart from

- the excitation in order to establish the QTEM mode
- any disturbance such as an impedance step at the port connection point

The port reference plane of usual ports in Empire is defined at the voltage and current recording location. For calibrated ports the reference plane may be shifted apart from the recording location by applying a line transformation. So, if the port reference plane has to be placed at the end of the port a calibrated port is recommended.

Hint: Calibrated Ports

- The reference plane can be shifted to an arbitrary location
- A voltage source may be used optionally
- No source area needed in case of Absorbing Port (fast simulation)
- A short port length is recommended since separation of incident and reflected waves is not accurate for multiples of half wavelength.

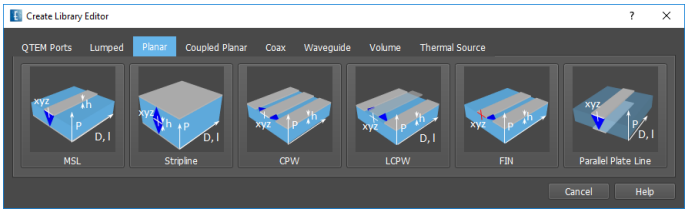


Figure 20.6.: Planar port types

Common Port Parameters

- While concentrated port types may be defined anywhere inside the simulation domain, absorbing port types have to be placed at a boundary with absorbing boundary conditions.
- Depending on the type of the port the reference plane (indicated by a blue line) is either at the boundary (absorbing port) or within the structure (concentrated port)
- The port length should be large enough to suppress contributions of evanescent modes

1. **Port Definition**
   - **Port Type**, $p1t$ **Concentrated** - off boundary, **Absorbing** - at boundary, **None** - strip only
   - **Port Number**, $p1 \geq 1$, to be adjusted for multiple ports
   - **Port Excitation** $p1e = On/Off$, **Off** - only used for termination
   - **Calibrated Port** $p2cal$, allows arbitrary reference plane shifts
   - **Reference Plane Distance** $p1ref$ distance from base pont in units. Position of reference plane (for calibrated ports, only)
   - **Measurement Distance** $p1d > 1cell$ distance from base pont in units. Position of voltage and current recordings(for calibrated ports, only)
2. **Source Type**
   - **Use series voltage source** (For calibrated ports=On, only)
3. **Concentrated Port Parameters** (only for port type = concentrated)
   - **Port Load Impedance**, $R > 0$ in Ω, internal load of current source
   - **Use Internal Port Impedance as Reference Impedance** $Rcopy = On/Off$. If set to **On** this value will be used for S-Parameters, see **Hint: Reference Impedance** in 20.1
   - **Port Measurement Distance**, $pd > 1cell$ in units, distance between load and reference plane
4. **Advanced Parameters**

- **Excitation Delay** $ED \geq 0$ in s or as abbreviation `angle@frequency`, used for narrow band phase shift in multiple simultaneous excitations
- **Broadband Phase Shift (AC only)** $PS \geq 0$ in $degree$, used for broadband phase shift in multiple simultaneous excitations (only possible for modulated pulses)
- **Excitation weight factor**, WE, used for weighting multiple ports
- **Current weight factor**, WC, used for, e.g. magnetic symmetry plane
- **Voltage weight factor**, WV, used for, e.g. electric symmetry plane

5. **Impedance Calculator**
   - **Rel. Permittivtiy** $epsr \geq 1$ value used for impedance formula in **Info** window

Planar Ports

After creation, the ports can be edited with **Open/Edit Object** 

Object Editor

- **Name** Optional name for display in group
- **Group** Group used for port
- **Plane** perpendicular to height (value = strip thickness)
- **Direction** of wave propagation (value = length of port)
- **Base Point** x,y,z define the start coordinates (location of load and excitation)

Remarks

- Ground metal (microstrip and stripline) and substrate have to be defined separately
- The port has to be defined on a group with **conductor** or **metal** property
- Using **Edit Settings** Port Parameters can be accessed

Planar - Microstrip Port

The input method is to define several points

1. Select plane or grid of the port plane
2. End point of port (Feed position)
3. Start point of port (Location of load)
4. Strip height
5. Strip width
6. Location of ground plane

Individual Port Parameters

- **Geometry** to be entered in drawing units.
  - **Width of Conductor**, $w > 0$
  - **Bottom height**, $|hb| > 0$, distance between signal line and ground
  - **Met Thickness**, $|t| > 0$, if **auto** the height h is used

  - **Upside Down** To create this port in reverse order, e.g. below ground
  - **Info** Depending on the current values the impedance is calculated with a formula.

Planar - Stripline Port

The input method is to define several points

1. Select plane or grid of the port plane
2. End point of port (Feed position)
3. Start point of port (Location of load)
4. Strip height
5. Strip width
6. Location of first ground plane
7. Location of second ground plane

Individual Port Parameters

- **Geometry** to be entered in drawing units.
  - **Width of Conductor**, $w > 0$
  - **Top height**, $ht > 0$, distance to top ground
  - **Bottom height**, $hb > 0$, distance to bottom ground
  - **Met Thickness**, $t > 0$, if **auto** the strip height is used
  - **Info** Depending on the current values the impedance is calculated with a formula.

Planar - Coplanar Port

The input method is to define several points

1. Select plane or grid of the port plane
2. End point of port (Feed position)
3. Start point of port (Location of load)
4. Strip height
5. Center strip width
6. Inner edge of outer conductor

Individual Port Parameters

- **Geometry** to be entered in drawing units
  - **Width of CPW Center Conductor**, $w > 0$
  - **Width of CPW Gap**, $s > 0$
  - **Width of CPW Outer Conductors**, $b > 0$
  - **Met Thickness**, $t > 0$, if **auto** the strip height is used
  - **Info** Depending on the current values the impedance is calculated with a formula

- **Concentrated Port Parameters** (only for port type = concentrated)
  - **Port Load size**, the length of the load in propagation direction

Planar - Lifted Coplanar Port

The input method is to define several points

1. Select plane or grid of the port plane
2. End point of port (Feed position)
3. Start point of port (Location of load)
4. Strip height
5. Center strip width
6. Inner edge of outer conductor

Individual Port Parameters

- **Geometry** to be entered in drawing units
  - **Width of CPW Center Conductor**, $w > 0$
  - **Width of CPW Gap**, $s > 0$
  - **Width of CPW Outer Conductors**, $b > 0$
  - **Lift of Center Conductor** l, defines the offset hight of center conductor with respect to outer conductors
  - **Met Thickness**, $t > 0$, if **auto** the strip height is used
  - **Info** Depending on the current values the impedance is calculated with a formula
- **Concentrated Port Parameters** (only for port type = concentrated)
  - **Port Load size**, the length of the load in propagation direction

Planar - Finline Port (Asymmetric Coplanar Port)

The input method is to define several points

1. Select plane or grid of the port plane
2. End point of port (Feed position)
3. Start point of port (Location of load)
4. Strip height
5. Center strip width
6. Inner edge of outer conductor

Individual Port Parameters

- **Geometry** to be entered in drawing units
  - **Width of Center Conductor**, $w > 0$
  - **Width of Gap1**, $s1 > 0$ right-handed side
  - **Width of Gap2**, $s2 > 0$ left-handed side

  - **Width of Outer Conductor 1**, $b1 \geq 0$, right-handed side
  - **Width of Outer Conductor 2**, $b2 \geq 0$, left-handed side
  - **Met Thickness**, $t > 0$, if **auto** the layer height is used
  - **Info** Depending on the current values the impedance is calculated with a formula.
- **Concentrated Port Parameters** (only for port type = concentrated)
  - **Port Load size**, the length of the load in propagation direction

Planar - Parallel Plate Line Port

The input method is to define several points

1. Select plane or grid of the port plane
2. End point of port (Feed position)
3. Start point of port (Location of load)
4. Strip height
5. Strip width
6. Distance to ground plane

Individual Port Parameters

- **Geometry** to be entered in drawing units
  - **Width of Conductor**, $w > 0$
  - **Height**, $h > 0$ Distance between conductors
  - **Met Thickness**, $t > 0$, if **auto** the layer height is used
  - **Info** Depending on the current values the impedance is calculated with a formula.

Coupled Microstrip Port

The input method is to define several points

1. Select plane or grid of the port plane
2. End point of port (Feed position)
3. Start point of port (Location of load)
4. Strip height
5. Strip width
6. Inner edge of second conductor
7. Distance to ground plane

Individual Port Parameters

- **Geometry** to be entered in drawing units.
  - **Width of Conductor**, $w > 0$
  - **Slot Width**, $s > 0$

  - **Bottom height**, $|hb| > 0$, distance between signal line and ground
  - **Met Thickness**, $|t| > 0$, if **auto** the height h is used
  - **Info** Depending on the current values the impedance is calculated with a formula
- **Advanced Parameters**
  - **Mode** Odd or even mode selection

Coupled Stripline Port

The input method is to define several points

1. Select plane or grid of the port plane
2. End point of port (Feed position)
3. Start point of port (Location of load)
4. Strip height
5. Strip width
6. Inner edge of second conductor
7. Distance to first ground plane
8. Distance to second ground plane

Individual Port Parameters

- **Geometry** to be entered in drawing units.
  - **Width of Conductor**, $w > 0$
  - **Slot Width**, $s > 0$
  - **Top height**, $ht > 0$, distance to top ground
  - **Bottom height**, $hb > 0$, distance to bottom ground
  - **Met Thickness**, $t > 0$, if **auto** the strip height is used
- **Advanced Parameters**
  - **Mode** Odd or even mode selection

Coax - Circular Coaxial Port

- The direction determines the orientation of the port
- x,y,z define the start coordinates (location of load and excitation)
- l defines the length of the line
- The group property does not have effects

The input method is to define several points

1. Select plane or grid perpendicular to the cross section
2. End point of port (Feed position)
3. Start point of port (Termination position)
4. Point on outside of inner condutor
5. Point on inside of outer condutor

Parameters

1. The info window calculates the impedance depending on the entered values
2. **Geometry** to be entered in drawing units
   - **Inner Diameter**, $di > 0$, inner conductor, ideal metal
   - **Dielectric Diameter**, $dd > di$, isolator
   - **Outer Diameter**, $da > dd$, outer conductor, ideal metal
   - **Rod length**, $lr \geq 0$, additional length of inner conductor
3. **Material** - Permittivity of the insulator
   - **Permittivity(rel)**, $epsr \geq 1$
4. **Geometric Priority** to be adjusted if port intersects other objects
   - **Outer Priority**, $pra > 0$
   - **Dielectric Priority**, $prd > pra$
   - **Inner Priority**, $pri > pri$
5. **Concentrated Port Parameters** (only for port type = concentrated)
   - **Use old concentrated ports**, $OCP = on/off$, no end cap is used if set to **on**

Coax - Square coaxial Port

- The direction determines the orientation of the port
- x,y,z define the start coordinates (location of load and excitation)
- l defines the length of the line

The input method is to define several points

1. Select plane or grid perpendicular to the cross section
2. End point of port (Feed position)
3. Start point of port (Termination position)
4. Point on outside of inner condutor
5. Point on inside of outer condutor

Parameters

1. The info window calculates the impedance depending on the entered values
2. **Geometry** to be entered in drawing units
   - **Inner Diameter**, $di > 0$, inner conductor, ideal metal
   - **Dielectric Diameter**, $dd > di$, isolator
   - **Outer Diameter**, $da > dd$, outer conductor, ideal metal
   - **Rod length**, $lr \geq 0$, additional length of inner conductor
3. **Material** - Permittivity of the insulator
   - **Permittivity(rel)**, $epsr \geq 1$
4. **Geometric Priority** to be adjusted if port intersects other objects
   - **Outer Priority**, $pra > 0$
   - **Dielectric Priority**, $prd > pra$
   - **Inner Priority**, $pri > pri$
5. **Concentrated Port Parameters** (only for port type = concentrated)
   - **Use old concentrated ports**, $OCP = on/off$, no end cap is used if set to **on**

20.1.5. TE-TM Waveguide Ports

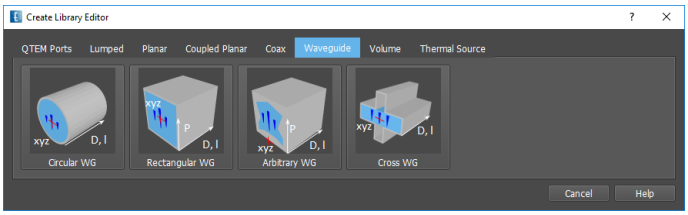


Figure 20.7.: Waveguide port types

The hollow waveguide ports

- Rectangular waveguide port
- Circular waveguide port
- Arbitrary waveguide port

are either concentrated or absorbing ports which are used to feed a waveguide and

- record voltage equivalent
- record current equivalent
- in case of concentrated port terminate with a load
- excite TE or TM modes if excitation is switched to **On**

Hint: Waveguides in lossy media

The mode excitation plane (defined by **Measurement Distance**) may not be placed inside lossy media.

- If waveguide ports are placed at lossy media it has to be ensured that the port length is longer than the **Measurement Distance**
- If waveguide ports are embedded in lossy media it is recommended to
  - set the permittivity of the lossy media in the **Material** parameter of the waveguide
  - set the priorities to a higher value than the lossy media in **Geometric Priority**

Common properties

- While concentrated port types may be defined anywhere inside the simulation domain, absorbing port types have to be placed at a boundary with absorbing boundary conditions.
- Depending on the type of the port the reference plane (indicated by a blue line) is either at the boundary (absorbing port) or within the structure (concentrated port)
- The group property does not have effects.
- The port length should be large enough to suppress contributions of evanescent modes

Common Parameters

1. **Port Definition**
   - **Port Type**, *p1t*
     - **Concentrated** Port apart from boundary

  - **Absorbing** Port placed at boundary
  - **None** only geometry (no excitation, load or recording)
- **Port Number**, $p1 \geq 1$, to be adjusted for multiple modes or ports
  - **Single number**, m Use mode with lowest cutoff frequency, only
  - **Number range**, $m - n$ Use first $(n - m + 1)$ TE and TM modes ordered by cutoff frequency
- **Port Excitation** $m \leq p1e \leq n$
  - **Off** - port only used for termination
  - **Single number**, q Excite with single mode
  - **Number range**, $q - p$ Excite with multiple $(p - q + 1)$ modes (sequential or simultaneous as set in **Simulation Mode**
- **Calibrated Port**, allows arbitrary reference plane shifts
- **Reference Plane Distance** position of reference plane (for calibrated ports, only)
- **Measurement Distance** position of excitation, voltage and current recordings (has to be less than port length)

2. **Material** - Dielectric filling of the waveguide
   - **Permittivity(rel)**, $epsr \geq 1$, used in simulation
3. **Concentrated Port Parameters** (only for port type = concentrated)
   - **Port Distance**, $pd > lsize$ in drawing unit, distance between waveguide start and reference plane
   - **Load Size**, $lsize > 1cell$, determines the thickness of the (PML-) load used for absorption
   - **Additional Load Rsquare in Ohms**, used to suppress cut-off modes in PML area
   - **Load min resolution (cells)**, number of cells used for PML area
4. **Advanced Parameters**
   - **Excitation Delay** $ED \geq 0$ in s or as abbreviation `angle@frequency`, used for narrow band phase shift in multiple simultaneous excitations
   - **Broadband Phase Shift (AC only)** $PS \geq 0$ in $degree$, used for broadband phase shift in multiple simultaneous excitations (only possible for modulated pulses)
   - **Excitation Weight Factor**, WE, used for weighting multiple ports
   - **Current Weight Factor**, WC, can be adjusted for special purposes
   - **Voltage Weight Factor**, WV, can be adjusted for special purposes
5. **Mode display** Create Mode previews on animation layers
   - **Field Components** Select field components to be displayed
   - **Layers** Specify single or multiple groups for display
   - **Separate Plots** Specify common or separate plots for different modes

Hint: Higher order modes

The number of higher order modes used as termination in the simulation can be set by setting the **Port Numbers** to **m-n** where m is the start port number and $n - m$ is the number of modes for this port to be taken into consideration. If all modes should be used as exciation this should also be entered as **Port Excitation**.
In case of simultaneous excitation and if a broadband phase shift has to be used, the phases in degree can be entered with a space charcater, e.g. **0 90** for the first 2 modes.

Circular Waveguide Port

Hint

This waveguide port supports dominant and higher order modes. If higher order modes have to be considered, the number can be specified in the Port Definition by, e.g. **1-3** as port numbers for the first three modes. Both polarizations of the dominant mode can be enabled by using **1-2** as port numbers.

- The Direction determines the orientation of the port
- x,y,z define the start coordinates
- l defines the length of the line
- The cut-off frequency of the dominant mode is displayed by pressing the **Info** button in the Parameters.

The input method is to define several points

1. Select plane or grid of the cross section
2. Center point
3. Radius
4. Length

Parameters

1. **Geometry** to be entered in drawing units.
   - **Inner Diameter** of the waveguide, $w > 0$
   - **Wall Thickness**, $dw > 0$
   - **Height Direction**, perpendicular to length
2. **Geometric Priority** - Only needed if port intersects with other objects
   - **Create waveguide parts** can be set to **Off** if waveguide walls already exist
   - **Outer Priority**, $10 \leq pra \leq 250$, priority of wall
   - **Dielectric Priority**, $pra \leq prd \leq 250$, priority of dielectric filling or air
3. **Mode Calculation**
   - **Analytic Modes (PGA only)** By default modes will be calculated with a 2D eigenvalue solver for the discretized waveguide cross section. If $amd = On$ modes will be calculated analytically and mapped to the grid. In this case the PGA option should be enabled.
   - **Mode tilt angle** If $amd = On$ the polarization angle can be chosen $0 \leq phi \leq 360$
   - **First Mode Number** Lower order modes will be neglected in simulation

Rectangular Waveguide Port

Hint

This waveguide port supports dominant and higher order modes. If higher order modes have to be considered, the number can be specified in Port Definition by, e.g. 1-3 as port numbers for the first three modes.

- Plane and Direction determine the orientation of the port
- x,y,z define the start coordinates
- l defines the length of the line
- The lower side of the internal waveguide wall is positioned at the group start height
- The cut-off frequency of the dominant mode is displayed by pressing the **Info** button in the Parameters.

The input method is to define several points

1. Select plane or grid of the cross section
2. Point 1 of the cross section
3. Point 2 of the cross section
4. Length

Parameters

1. **Geometry** to be entered in drawing units.
   - Inner **Width** of the waveguide, $w > 0$
   - Inner **Height** of the waveguide, $h > 0$,
   - **Wall Thickness**, $dw > 0$
2. **Geometric Priority** - Only needed if port intersects with other objects
   - **Create waveguide parts** can be set to **Off** if waveguide walls already exist
   - **Outer Priority**, $10 \leq pra \leq 250$, priority of wall
   - **Dielectric Priority**, $pra \leq prd \leq 250$, priority of dielectric filling or air

Arbitrary Waveguide Port

This port supports arbitrary shaped waveguide cross section. The geometry has to be defined separately, e.g. by drawing a tube with dielectric filling which represents the cross section. A background conductor with lower priority can be defined in the parameters.

Hint

This waveguide port supports dominant and higher order modes. If higher order modes have to be considered, the number can be specified in the Port Definition by, e.g. 1-3 for the first three modes.

- Plane and Direction determine the orientation of the port
- x,y,z define the start coordinates
- l defines the length of the line

- The lower side of the internal waveguide wall is positioned at the group start height
- The cut-off frequency of the modes are unknown until the simulation starts and listed in the simulation log window.

The input method is to define several points

1. Select plane or grid of the cross section
2. Point 1 of the cross section
3. Point 2 of the cross section
4. Length

Parameters

1. **Geometry** to be entered in drawing units.
   - **Width** of the total cross section, $w > 0$
   - **Height** of the total cross section, $h > 0$
2. **Background conductor** - Outer metal filling of the waveguide
   - **Create background conductor**, $box = On/Off$, creates a metal surrounding optionally
   - **Conductor Priority**, $10 < pra < 250$, specifies geometric priority of metal surrounding

20.2. Volume Sources

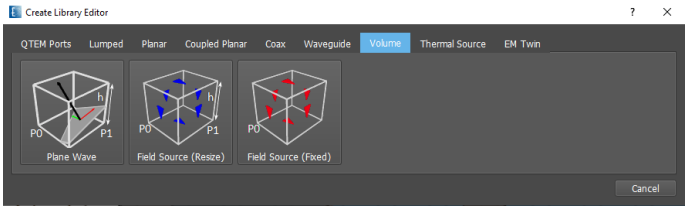


Figure 20.8.: Volume sources

20.2.1. Plane Wave

Plane waves are realized as compact sources which drive magnetic and electric surface currents on a box surface defined by 2 Points and height. This means that inside the box both the incident field wave as well as the scattered field exists, whereas outside the box only the scattered field is present.

All scattering objects must lie inside the plane wave box. The scattering objects may not touch or intersect the box. The surrounding has to be air. The boundary may be chosen to be electric or magnetic if the polarization is chosen accordingly.

With **Edit Settings** the properties of the plane wave have to be defined.

The direction of propagation is entered in spherical co-ordinates in degrees. The polarization of the wave is uniquely defined by components of the spherical unit vectors.

This table shows the corresponding spherical coordinates for the main axes directions:

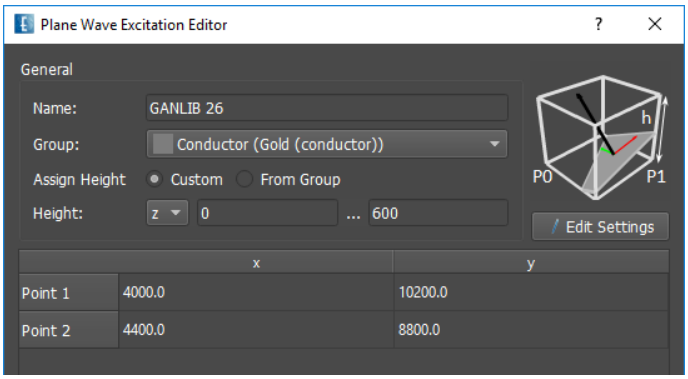


Figure 20.9.: Parameters of the plane wave

direction	polarization	k_θ	k_φ	E_θ	E_φ
$\hat{x}$	$\hat{y}$	90°	0	0	E_0
$\hat{x}$	$\hat{z}$	90°	0	$-E_0$	0
$\hat{y}$	$\hat{x}$	90°	90°	0	$-E_0$
$\hat{y}$	$\hat{z}$	90°	90°	$-E_0$	0
$\hat{z}$	$\hat{x}$	0	0	E_0	0
$\hat{z}$	$\hat{y}$	0	90°	E_0	0

For general direction and polarization the formula given in 44.2 can be applied.

More than one plane wave box may be defined to exploit interference effects, e.g. a standing wave to model a homogeneous field. To calculate delay factors one plane wave has to be regarded as reference.

- **Direction**
  - **K_theta** - θ- component for direction of propagation in **degrees**
  - **K_phi** - ϕ- component for direction of propagation in **degrees**
- **Field Strength**
  - **E_theta** - θ- component of electric field for polarization in $\frac{V}{m}$
  - **E_phi** - ϕ- component of electric field for polarization in $\frac{V}{m}$
- **Options**
  - **Port Number**, used for multiple excitations and result folder number
  - **Enable Plane Wave**, can be switched off for special purposes
  - **Dispersion Correction**, accuracy improvement for large structures
  - **Delay** in **s**, the excitation may be delayed against other ports or sources
  - **Broad Band Phase Shift** in **degrees**, phase differnce against other ports or sources (AC only)
  - **Disable Sides**, sides can be switched off for special purposes
  - **Display Plane Position** in percent. Used for display of the plane wave symbol only.
  - **Phase Reference Position** in units. Excitation reference plane

In 2D, 3D Design and Result Tabs the plane wave is represented by a box with the propagation direction (orange arrow) electric field direction (red arrow) and magnetic field direction (green arrow).

20.2.2. Field Source (Resizable)

Field Sources are volume field dumps in frequency domain obtained from a simulation or external data read from a file. They are realized as compact sources which drive magnetic and electric surface currents on a box surface defined by 2 Points and height.

An external data file has to be in ASCII e.g. **.csv** format and contain x,y,z positions in m and the respective field component, e.g. Hx,Hy,Hz. If only one field strength is specified, the simulation calculates the missing field strength by solving a system of equations.

The field dump used from a previous simulation has to be in ASCII format **.dat**, too. This option can be specified with a right click on field monitor and select **Export Raw Data (Full)**.

The typical workflow is as follows:

Simulate Antenna

- Record E & H at Target Frequency
- Use field dump size large enough (min distance to antenna lambda/4)
- Export E & H ASCII .dat files (Export Raw Data in 3D Results)
- Copy E.dat, H.dat to Field Source simulation folder

Simulate Source

- Source Volume should be smaller than dump volume (e.g. 1 cell smaller) if **H-Excitation Location** is set to **outside**
- Use both E & H volume .dat files for excitation
- Apply meshhint for source: Edge use: all, Refinement 3...5 (used 4)
- Enable Modulated Pulse
- Use small bandwidth (Start & Stop Frequency close to Target Frequency)
- Time Step Limit large enough to account for broader pulse

If the support points don't coincide with the current grid, an interpolation scheme is used to excite the surface current on each surface cell.

Because the sources are valid for a single frequency, only, the simulation should be carried out using a modulated Gaussian pulse with the center frequency of the recorded frequency (e.g. using the **AC only** option in the **Simulation Setup**

With **Edit Settings** the properties of the field source can be defined.

- **Source**
  - **E-Field File**, data of the E-field, supported formats **.dat**, **.csv**
  - **H-Field File**, data of the H-field, supported formats **.dat**, **.csv**
- **General**
  - **Port Number** can be specified if multiple excitations are used
  - **H-Excitation Location**, select if the H-field excitation should be located a half cell shifted to **inside** or **outside** of the field source volume

  - **Disable Sides**, single sides may be switched off for special purposes
  - **Delay**, the excitation may be delayed against other ports or sources
- **Corner Coordinates** Minimum corner coordinates of original field dump
  - **x in m** - coordinate shift of source related to original position in x direction
  - **y in m** - coordinate shift of source related to original position in y direction
  - **z in m** - coordinate shift of source related to original position in z direction
- **Options**
  - **E-Field Factor**, weight for E-Field
  - **H-Field Factor**, weight for H-Field

Format Convention for .csv file

- Values separated by semicolons
- First row contains descripters

Minimal .csv Example (400mm cube)

X[m];	Y[m];	Z[m];	|Hx[A/m]|;	∠ Hx[°];	|Hy[A/m]|;	∠ Hy[°];	|Hz[A/m]|;	∠ Hz[°]
-0.2;	-0.2;	-0.2;	1;	0;	1;	90;	0;	0
-0.2;	-0.2;	0.2;	1;	0;	1;	90;	0;	0
-0.2;	0.2;	-0.2;	1;	0;	1;	90;	0;	0
-0.2;	0.2;	0.2;	1;	0;	1;	90;	0;	0
0.2;	-0.2;	-0.2;	1;	0;	1;	90;	0;	0
0.2;	-0.2;	0.2;	1;	0;	1;	90;	0;	0
0.2;	0.2;	-0.2;	1;	0;	1;	90;	0;	0
0.2;	0.2;	0.2;	1;	0;	1;	90;	0;	0

20.2.3. Field Source (Fixed Size)

This Field Source in frequency domain is obtained from external data read from a file (Rohde&Schwarz (R+S) Format). It is realized as compact source which drive magnetic and electric surface currents on a box surface. The dimensions of the box are defined by the external data, so it has a fixed size. The Point of the Field Source shifts/moves the box relative to its original position.

The (primary) external data in R+S Format can be created from measurement data as described in [2]. It needs to be converted into the (secondary) external data in E+H-Field Surface File Format (.surf.dat), which can then be used by the simulation kernel to excite the fields.

- **Primary external data**, R+S Format, not directly usable by simulation kernel, consisting of three files:
  - ***.hmj file**, Altair Hypermesh Generic ASCII Format, Electric Surface Currents in CTRIAOB
  - ***.hmm file**, Altair Hypermesh Generic ASCII Format, Magnetic Surface Currents in CTRIAOB
  - ***.nas file**, NASTRAN CAD file. Must describe a single cartesian box geometry. Must only contain triangles (CTRIA3). Number of triangles must exactly match the number of Electric/Magnetic Surface Current datasets.
- **Secondary external data**, .surf.dat Format, created by conversion from R+S Format, directly usable by simulation kernel, consisting of a single *.surf.dat file

Because the sources are valid for a single frequency only, the simulation should be carried out using a modulated Gaussian pulse with the center frequency of the recorded frequency (e.g. using the **AC only** option in the **Simulation Setup**

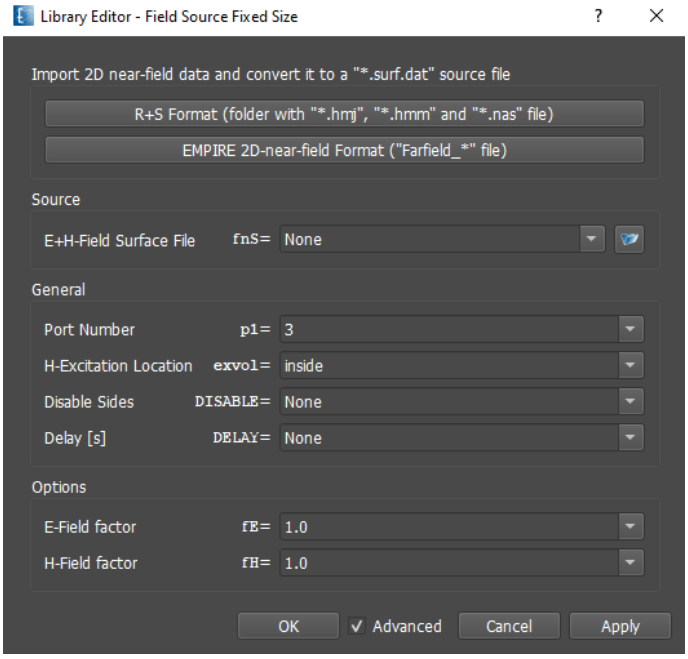


Figure 20.10.: Parameters of the fixed-size field source

With **Edit Settings** the properties of the field source can be defined.

- **Source**
  - **E+H-Field Surface File**, data of the E and H-field
  - **Import Current based 2D Nearfield Data (R+S Format)**, this button starts the conversion from primary (R+S Format) to secondary (NF SURF Format) external data
- **General**
  - **Excitation Volume**, either **inside** or **outside**
  - **Disable Sides**, single sides may be switched off for special purposes
  - **Delay**, the excitation may be delayed against other ports or sources
- **Options**
  - **E-Field Factor**, weight for E-Field
  - **H-Field Factor**, weight for H-Field

20.3. Thermal Source

In order to calculate a temperature distribution, thermal sources have to be defined. Depending on the chosen Solver type in the simulation setup the following sources can be used.

- **THERMAL** Solver requires Thermal Power source

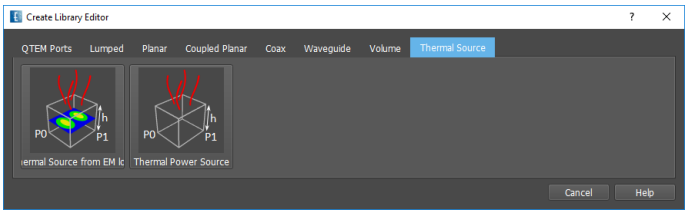


Figure 20.11.: Thermal sources

- **EM+THERMAL** Solver requires EM loss thermal source

20.3.1. Thermal Source from EM loss

In a combined EM and Thermal simulation, losses are calculated in an EM simulation which are used as a source for a subsequent thermal simulation. This source is specified by a volume defined by 2 Points and a height. This box should cover the lossy objects, e.g. dielectric with loss tangent, resistive conductors or body models.

The electromagnetic field is recorded for a specified frequency and the dissipated power will be calculated. The power distribution will then be used for the thermal simulation.

Thermal EM Source Parameters

- **General**
  - **Dump Box File name** Should be unique (if more than one source is used)
  - **Frequency** Actual frequency for calculating the losses
  - **Target frequency of LF-Algorithm** if Low Frequency Algorithm should be applied
- **Normalization**
  - **Normalization** to current (1A), voltage (1V), power (1W), or excitation, e.g. plane wave (1V/m)
  - **Weight Factor** additional scaling factor for the calculated losses
- **Switches**
  - **Loss Calculation Method** The power loss can be calculated from electric field and conductivity or from integration of the Poynting Vector

20.3.2. Thermal Power Source

In a pure thermal simulation sources can be defined by specifying a thermal source with a given power loss. This method is well suited for amplifiers, resistors, LEDs, etc. when their losses are known. The source is specified by a volume defined by 2 Points and a height. The power distribution of this volume is assumed to be equal.

Thermal Power Source Parameters

- **General**
  - **Thermal Power** Loss power of the element
  - **Thermal Conductivity** Conductivity of the source (should be high-valued for a uniform distribution of the power)

- **Display**
  - **Label** A label can be defined for display

20.4. EM Twin

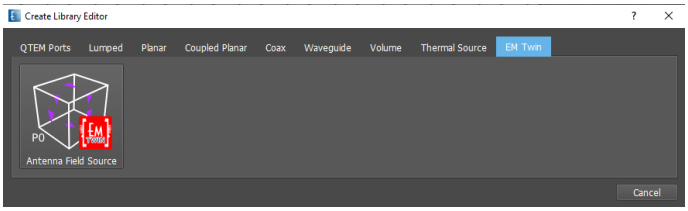


Figure 20.12.: Thermal sources

Create an **Antenna Field Source (AFS)** in order to excite with a near field antenna digital twin. See **Getting Started EM-Twin.pdf** for details.

21. Variables

Variables are used to parametrically define sizes or properties of objects or simulation parameters. Variables in **2D** or **3D-Design** Tabs can be used for **Sweeps** or **Optimizations** in EM simulations while Variables in **Circuit** Tab can be used during Circuit Simulation

Several types of variables can be defined:

- **Constant** Variable assigned to a real value
- **Stepped** Variable assigned to a range of real values with step size. A slider is available to adjust the current value.
- **Count** Variable assigned to a range of real values with number of values. A slider is available to adjust the current value.
- **List** Variable assigned to values contained in a list separated by commas as shown in Fig.21.1. A slider is available to adjust the current value.
- **Equation** Variable assigned to an equation which may contain other variables

Variables are listed in the left tree and can be edited with right click on their name.

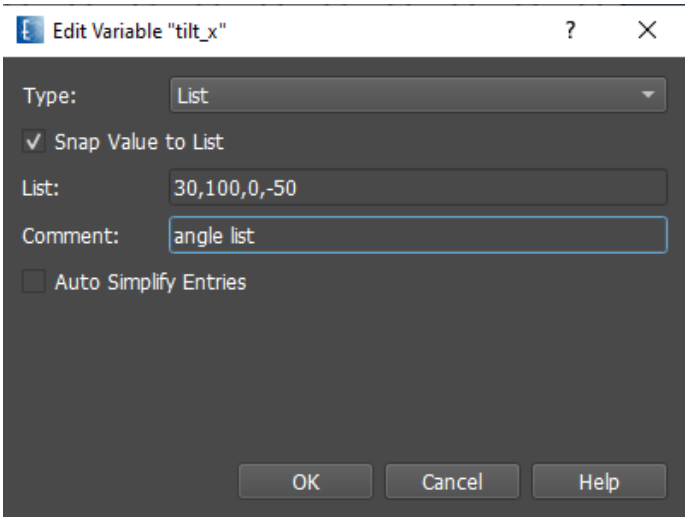


Figure 21.1.: Variable as list

Common properties of variables are

- **Name** Identifier as listed in the left tree and in folder names of sweeps or optimizations
- **Comment** Description of the variable as appears when hovering over the variable name
- **Type** One of the type described above

Hint

The name of the variable may not contain special characters and may not be a Python function name such as **max sin exp,...**. In case of many variables it is recommended to use short names, e.g. 2 characters, because they will be part of the folder names which are created for each variable set.

21.1. Create Variables

If no variable exists or a new variable should be created they can be created before usage by right click on **Variables - Create New Variable.**

Furher, a new variable is created as soon as a new variable name is entered in an input field.

A small editor window is opened where name, comment, and type can be entered. Depending on the type values can be edited. A switch can be used to simplify the entered terms.

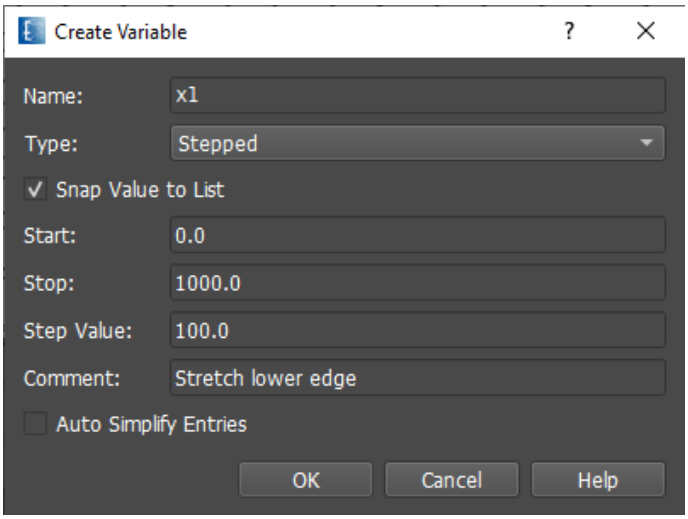


Figure 21.2.: EMPIRE XPU™ Variable editor

21.2. Control Variables

Right click on **Variables** opens the control of all variables

- **Create New Variable** Open the variable editor
- **Sort Variables** Sort variable according to name or type
- **Delete - Unused**: Remove all parameters which are currently not in use
- **Delete - All**: Remove all parameters and replace them by their current values
- **Lock All** : Prevent parameters against slider misadjustments or overwriting during import
- **Unlock All** : Allow parameter slider for modification of current value
- **Save Settings**: Store all parameters into a single file **.pms**

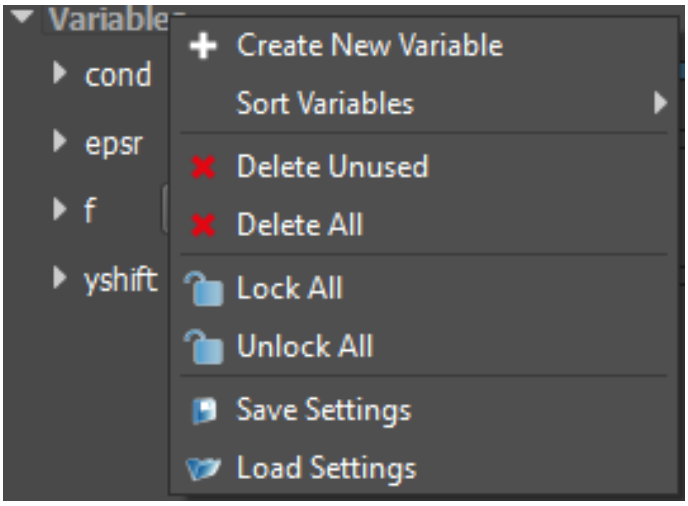


Figure 21.3.: EMPIRE XPU™ All variables control

- **Load Settings**: Read file **.pms** and restore all name matching parameters

Right click on **Variable Name** opens the control of single variables

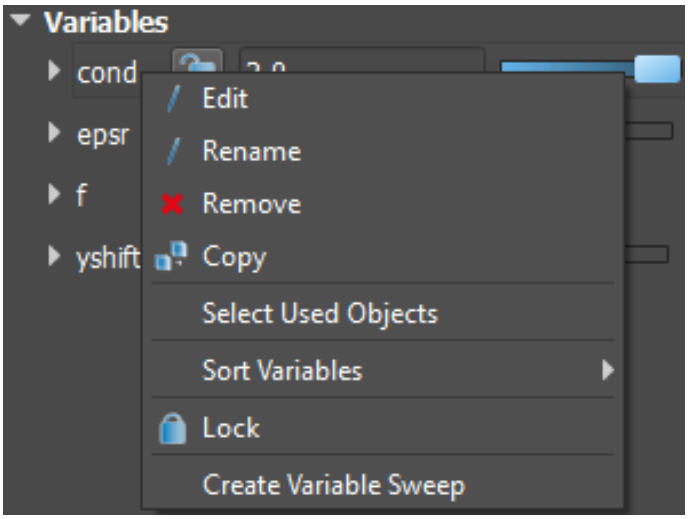


Figure 21.4.: EMPIRE XPU™ Single variable control

- **Edit**: Open the variables editor
- **Rename**: Assign a new name for this parameter
- **Remove**: Delete this variable from all objects and entries
- **Copy**: Copy this variable to a new one
- **Select Used Objects**: List of objects which are using one of the parameters defined below
- **Sort Variables**: Sort variable according to name or type

- **Lock**: Prevent variable from modification

With a left click on ▶ **Variable Name** the properties of the variable is listed. Values can be adjusted with a double click on the respective entry

Remarks

- Automatic variable sweeps are very convenient and can drastically reduce the work compared to a manual setup of the simulation jobs. Care has to be taken that the amount of data does not explode. It is recommended to deselect unused variables for sweep/optimization. See 24.7 for an estimation of parameter combinations which are possible.
- Not only the amount of variables should be considered but also the step size and how many intervals will be generated for each variable in order not to produce too much data. It is recommended to use large step sizes if possible.
- Each variable combination creates a new result folder with a complete set of result files including near and far field dumps. It is recommended to deactivate field recordings if possible.
- In case of multi-port simulation multiple subsequent simulations are generated for each variable sweep.
- The sweep result folder names contain variable name and current value. Because the folder name is limited in number of characters, short variable names should be preferred.
- If Automatic meshing is not disabled a new mesh is generated for each variable sweep.

21.3. Equations

Equations in EMPIRE XPU™ are mathematical expressions that typically contain intrinsic functions, arithmetic operators and defined variables. Equations cannot be directly subject of sweep or optimization.

The following mathematical functions may be used to define expressions:

	Syntax	Description
Intrinsic functions	$exp(x)$	Return the exponential function of x, e^x
	$log(x)$	Return the natural logarithm of x, $log_e(x)$
	$log10(x)$	Return the base-10 logarithm of x, $log_{10}(x)$
	$sqrt(x)$	Return the square root of x, $\sqrt{x}$
	$abs(x)$	Return the absolute value of x, $\|x\|$
	$hypot(x, y)$	Return the Euclidean norm, $\sqrt{(x^2 + y^2)}$
	$degrees(x)$	Convert angle x from radians to degrees
	$radians(x)$	Convert angle x from radians to degrees
Trigonometric functions	$acos(x)$	Return the arc cosine of x, in radians
	$asin(x)$	Return the arc sine of x, in radians
	$atan(x)$	Return the arc tangent of x, in radians
	$cos(x)$	Return the cosine of x, in radians
	$sin(x)$	Return the sine of x, in radians
	$tan(x)$	Return the tangent of x, in radians
	$acosh(x)$	Return the inverse hyperbolic cosine of x
	$asinh(x)$	Return the inverse hyperbolic sine of x
	$atanh(x)$	Return the inverse hyperbolic tangent of x
	$cosh(x)$	Return the inverse hyperbolic cosine of x
	$sinh(x)$	Return the inverse hyperbolic sine of x
	$tanh(x)$	Return the inverse hyperbolic tangent of x
Pre-defined Constant	pi	Return $\pi = 3.14159265358979$
	c_0	Return the speed of light in vacuum (m/s), 299792458

Table 21.1.: Mathematical functions

In EMPIRE XPU™ some pre-defined constants are available. These cannot be used to define a parameter and may not be reassigned a new value.

As a special equation the index of a list can be defined as variable to be used in a sweep or optimization as shown in Fig. 21.5

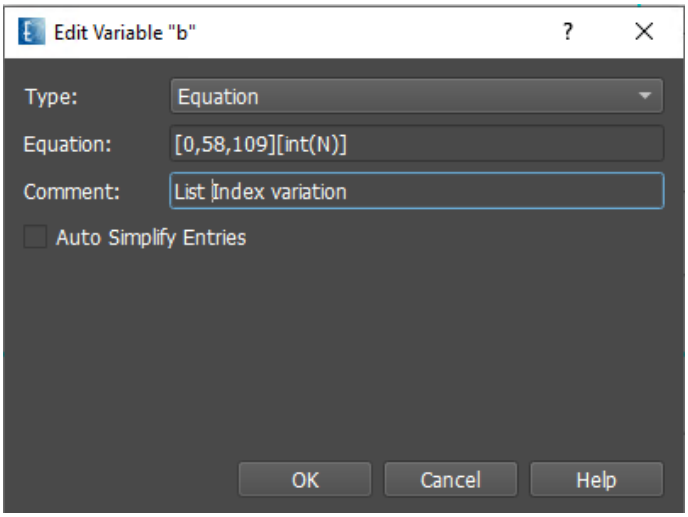


Figure 21.5.: Variable N as list index

21.4. Sweep Simulation

If Variables of type **Stepped, Count, List** have been defined and they are listed in the **Variables** entry in the left side menu they may be used for automatic sweep 24.7 and optimization 24.9 simulations in the simulation tab.

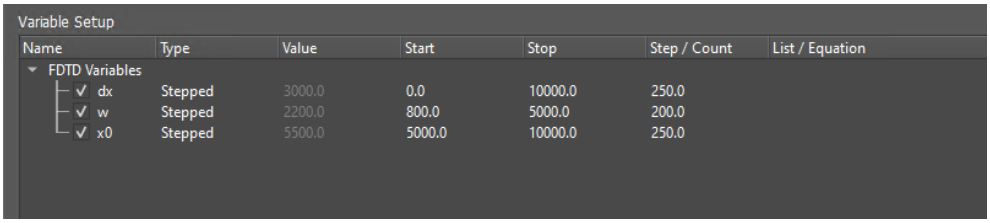


Figure 21.6.: Variable setup in Sweep and Optimization setup

For each variable set a new subfolder will be created and simulations will be carried out sequentially (on a single machine) or parallel (on a cluster with multiple installations).

21.5. Sweep Results

If sweep or optimization simulations are finished their results can be displayed in the respective **2D Results** Tab 25. The sweep parameters will be listed in separate columns in the Legend section and curves can be sorted in descending or ascending order or filters can be applied to reduce the number of entries.

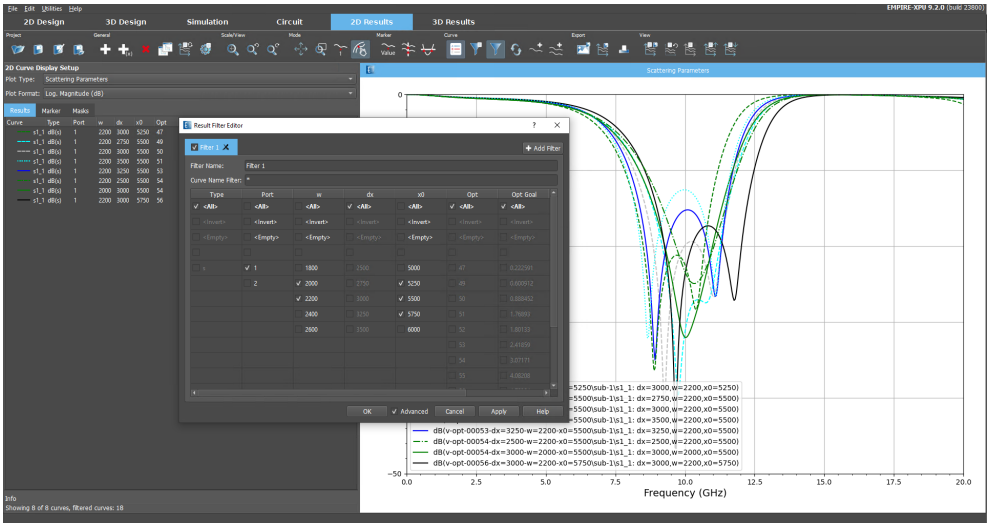


Figure 21.7.: Variable control for result curve display

22. Meshing and Simulation Domain

Objects and Sources entered in the **2D Design** or **3D Design** Tab determine the size of the simulation domain. At the borders of the simulation domain **Boundary Conditions** are applied. If required additional space can be added between the overall bounding box of objects and sources and boundary.

Inside the simulation domain the space is subdivided by a rectangular mesh which cell size is determined by meshing rules such as minimum resolution of wavelength or structure details. The total number of mesh cells determines the required memory for simulation. The size of the smallest mesh cell defines the stable time step value for the FDTD algorithm.

After creation the mesh can be visualized by:

- **3D Design** Right click to open the context menu and checkmark **Grid - Show Mesh**. Use space bar to toggle for different planes.
- **2D Design** Mesh bars on right and bottom (mesh lines can be visualized in Editor Options - Display Mesh)
- **3D Results** Press button **Mesh visualization** to obtain a menu with sliders to scroll the mesh lines
- **Meshed 3D Structure** Press button to show the discretized structure in the 3D Results Tab. Switch off groups to hide parts to improve display if necessary.

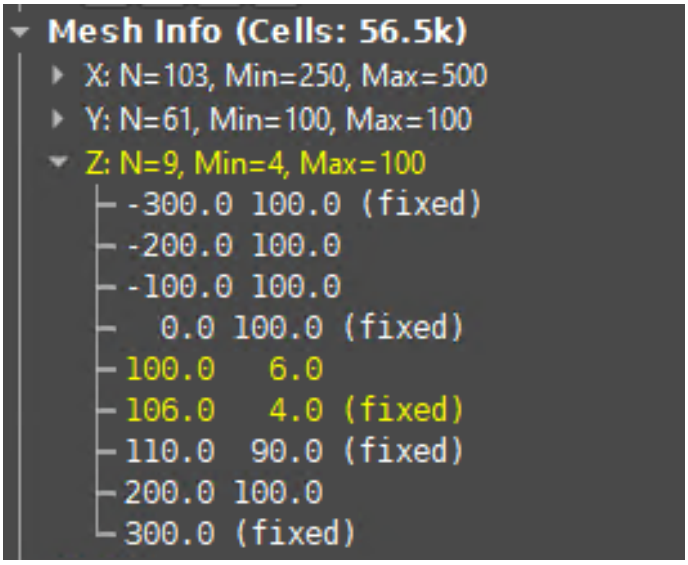


Figure 22.1.: Mesh information in navigation tree

The mesh information **Mesh Info** is available in the navigation tree, see Fig.22.1. It shows the number, minimum and maximum spacing of the cells for each direction. If the ratio between maximum and minimum cell spacing is larger than a certain value the entry will be colored.

- **white** Ratio is below 10

- **yellow** Ratio is between 10 and 100
- **red** Ratio is over 100

Small cell spacing lead to small timesteps and long simulation times. In some cases this can be avoided by setting mesh hints. The respective direction can be opened and the mesh line position and spacing to the neighbor line is displayed. This helps to detect the location of the smallest cell.

Hint: Application Note

More information is available in Chapter 29.2.14

22.1. Automatic Meshing

22.1.1. Mesh Creation

The automatic meshing creates a suitable mesh for the entered structure. It can be executed at any time e.g. to check the meshing of the current structure, and is executed automatically before a simulation is started if the **Mode** is not set to **Manual**.

If only one direction should be remeshed, e.g. after a change in the drawing it can be done with a right mouse click on the respective mesh bar in the 2D Design Mode and selecting **Automesh**.

The automatic meshing detects all objects and inserts mesh lines (blue lines in 2D Design mode) at object's edges, corners or other important coordinates. If lines are too close a merging algorithm is applied. Finally a filling algorithm is applied if mesh lines are too far apart (red lines in 2D Design mode). Meshing hints for refinement or coarsening can be defined in the following ways:

- Set global mesh hints, see Section 22.1.4
- Switch the group mesh level, see Section 22.1.8
- Add group property with a mesh hint, see Section 18.5.1

22.1.2. Simulation Domain

The automatic meshing also defines the boundaries of the simulation domain by following certain rules coming from far field definition, structure type, port size, etc.

- Some ports require a minimum space (e.g. horizontal distance of a microstrip line) which determine the minimum simulation domain
- Antennas require a certain space to the boundary.
  - A far field monitor automatically adds a quarter wavelength (of minimum recording frequency) around the structure.
  - **Structure Type** = **Antenna** automatically adds a quarter wavelength (of End Frequency) around the structure.
- **Structure Type** = **Inductor** automatically adds space around the structure
- Additional Space can be added for each side by defining a Boundary Distance in the **Simulation Setup - Boundary**
- The outermost mesh lines can also be entered manually, by pressing the middle mouse button on a grid bar in the 2D Design mode

The outermost mesh lines define the boundaries of the structure. The mesh can be deleted for all directions by pressing **Advanced** **Mesh Delete Lines** button. For single directions the mesh lines can be deleted by a right mouse click on the respective mesh bar and selecting **Delete Mesh** in the menu.

Hint: Grid statistics

Information about the current mesh can be obtained by open the left tree ▸ **Mesh Info**

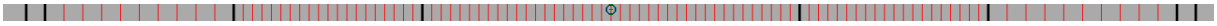


Figure 22.2.: 2D Design: Bottom mesh bar

In most cases the adjustments in the Simulation Setup 23.2.4 are sufficient for a suitable mesh. The following sections explain the mesh options in **Simulation Setup** - **Mesh** if the generated mesh should be modified.

22.1.3. General Mesh Options

In order to adjust the mesh options press the button and switch to the **Mesh** Tab.

- **Mesh Resolution** - Mesh accuracy (qualitative)

Resolution	Cells per wavelength N_λ	Cells for objects	Numeric value
Very coarse	8	3	-1
Coarse	10	3	1
Medium	15	4	5
Fine	20	5	10
Very Fine	25	6	15
Very Fine 2	28	7	20
Very Fine 3	31	8	25
Very Fine 4	34	9	30

- **Mode** - Algorithms can be optimized depending on information about the object types (mainly planar or mainly 3-dimensional)
  - **Planar + 3D** Here, the mesh will be optimized for both planar and general 3D structures. The planar plane is assumed to be xy-plane by default but can be set differently in the options menu.
  - **Planar** Mesh lines will be preferred for planar structure, over generic mesh lines from 3D objects.
  - **3D** Only 3D objects will be assumed and no optimization for flat metals will be performed.
  - **User Setup** Parameters of the automatic meshing algorithm in column **User Setup** can be adjusted only, if this mode is set.
  - **Manual** This mode is used to switch off the automatic meshing algorithm and to define the mesh manually.
- **Resolve Metal sheets**
  - **thin**, metallization will be resolved with one mesh line
  - **1 cell**, metallization will be resolved with two mesh lines with the spacing of the conductor thickness. This option should be used if the currents on top and bottom of a strip are significantly different.
- **Mesh Gaps to hard boundaries**
  - **off**, do not insert extra cells in gaps to hard boundaries
  - **electric**, insert extra cells in gaps to electric boundaries

  - **magnetic**, insert extra cells in gaps to magnetic boundaries
  - **on**, insert extra cells in gaps to electric and magnetic boundaries
- **Resolve Curved Geometry** Increases mesh resolution in areas with round geometries

22.1.4. Mesh Mode and Mesh Hint

Mostly used for defining a global mesh hint or for generating an equidistantly spaced mesh.

Mesh Mode

x,y,z-mesh in units

- **off** The automatic meshing is turned off in this direction.
- **auto** Automatic meshing will be used.
- **auto+fixed** As **auto** but fixed lines (black and bold) will be kept.
- **number** Equidistant mesh with a spacing of selected value.

Mesh Hint

x,y,z-mesh options in units (Global Mesh Hint)

- **off** No global mesh hint
- **min** Set the **Target Minimum**, limit the minimum cell size to the entered value, approximately
- **max** Limit the maximum cell size to the entered value, approximately
- **force_min** Limit the minimum cell size to the entered value, precisely
- **force_max** Limit the maximum cell size to the entered value, precisely
- **symmetry** Create a symmetric mesh with a symmetry plane at specified value

Minimum and maximum mesh hints can be combined, e.g. **min 100 max 1000**.

Hint: Target Minimum Cell Size

By default the target minimum cell size is derived from the mesh resolution and wall distance in each direction, see d_{min} in 22.1.9

- In some cases this value is too small causing very dense mesh which is not always necessary but results in long simulation runs. The Global Mesh Hint can help to increase the mesh spacing by setting the **min** value.
- In some cases this value is too large which can lead to detail neglection. The Global Mesh Hint can help to decrease the mesh spacing by setting the **max** value.

22.1.5. Simulation Box

In some cases the simulation domain should be truncated to model only a part of the structure, e.g. for large circuits where far distance components can be neglected.

In other cases it can be useful to fix the boundaries to prevent enlarging the domain in parameter sweeps or optimizations caused by the automatic meshing.

If the simulation domain should be fixed to certain values the **User defined Simulation Box** should be checkmarked in the **Simulation Setup - Mesh**

- The numbers displayed in **min** and **max** are the current boundary positions
- To adjust the simulation box by hand, the co-ordinates can be edited.

22.1.6. Boundary Distance

In some cases additional space to the boundaries will be required which can not be detected from the automatic meshing routines, e.g. if coils or antennas with a large reactive volume should be simulated. For each side, space can be added either in units, wavelength at End frequency or percentage of current simulation area see Fig.22.3

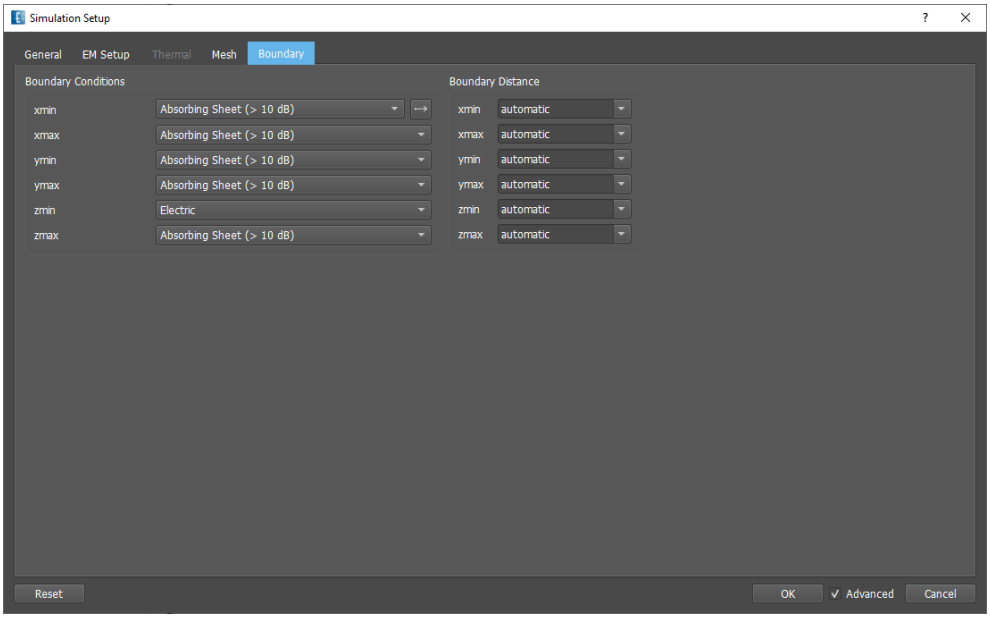


Figure 22.3.: Boundary Settings

If a boundary distance is already defined automatically (e.g. by far field monitor) the larger distance applies.

22.1.7. Advanced Mesh Setup

The options in the **Mesh** Tab can only be accessed if **Advanced** is checkmarked at the bottom.

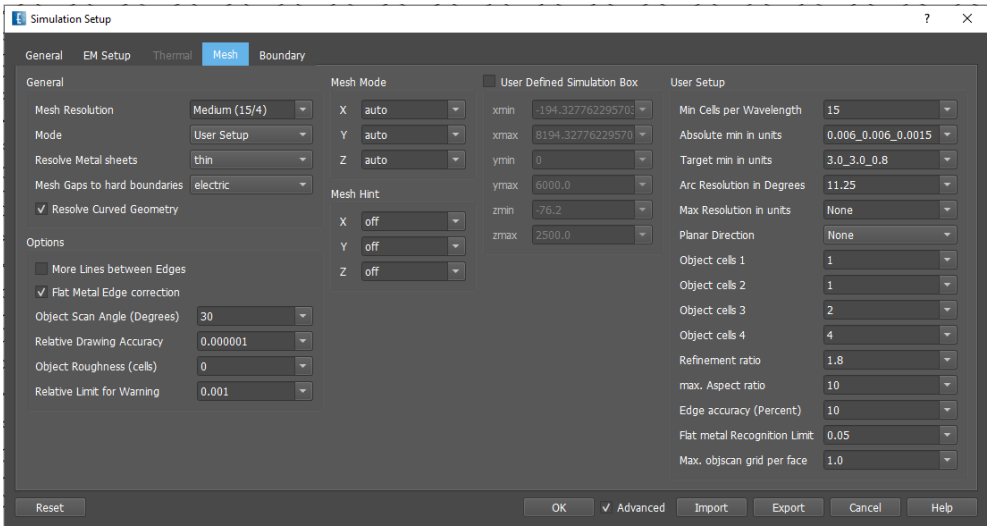


Figure 22.4.: Advanced Mesh Settings

Advanced Options

- **More Lines between Edges** Special algorithm for thin curved object
- **Flat Metal Edge correction** Enabled by default if PGA is ”off”, which accounts for edge effects on flat metal. The ”1/3rd - 2/3rd” rule is applied at flat metal edges.
- **Object Scan Angle (Degrees)** Defines a minimum angle of object segments and the scanning direction for mesh generation.
- **Relative Drawing accuracy** This number times the boundary distance gives the absolute drawing accuracy in each direction. Below this threshold co-ordinates will be treated as identical.
- **Object roughness (cells)** Value can be adjusted for improper drawings
- **Relative Limit for warning** A warning message will be displayed if mesh spacing is below this number times the boundary distance in each direction.

Advanced User Setup

Values can only be modified if **Simulation Setup - Mesh tab - General - Mode** is set to **User Setup**.

- **Min Cells per Wavelength** The minimum number of cells per wavelength may not be smaller than 6. Recommended 15. High precision 20.
- **Absolute min resolution in units** Distances below these numbers will be treated as equal.
- **Target min in units** The meshing engine tries to merge close lines if their spacing is below this limit.
- **Max. Resolution in units** Maximum cell spacing, if set to **None** it will be determined from maximum frequency.
- **Arc Resolution in Degrees** Angular step to discretize circles and arcs.
- **Planar Direction** Default value is z (xy-plane). Can be changed if needed.
- **Object cells** Number of cells used to resolve different objects
    1. number of cells for gaps or lumped elements

2. number of cells for objects on groups with meshing level 'Mesh objects edges only'
3. currently not in use
4. number of cells for objects on groups with meshing level 'Mesh objects edges and interior'

- **Refinement ratio** Used to generate a graded mesh between small and large cells.
- **max. Aspect ratio** Upper limit for ratios of cell length, height, width
- **Edge Accuracy in percent** Used for merging of grid lines (100% - no merging of grid lines)
- **Flat metal Recognition Limit** This factor is used to decide if metal is treated as flat.
- **Max. objscan grid per face** Maximum number of scan points a single object can create per face. Value should be enlarged if edges of objects could not be detected (can increase time needed for meshing)

22.1.8. Group Meshing

Meshing hints can also be given for each group separately by switching the button 

-  - Default mesh level, mesh objects edges and interior
  Examples: Signal conductors, dielectrics between signal and ground
-  - Mesh objects edges only
  Examples: Ground conductors, capacitive dielectric sheets, ground vias, lumped ports
-  - No meshing for objects
  Examples: Complex conductor objects with non-relevant details such as drill holes
-  - Mesh object edges and fine interior
  Examples: coplanar waveguide, slot line
-  - Ignore edges, interior only
  Examples: multiple objects with un-aligned edges
-  - Mesh gaps, only
  Examples: Feedthrus
-  - Bounding box, only (e.g. apply λ_{min}/N_λ for dielectric solids)
  Examples: Complex dielectric objects with non-relevant details such as drill holes

A more sophisticated mesh hint can be given for each group by adding the property **Mesh hint**, see 18.5.1

22.1.9. Meshing Algorithm

This section describes how mesh lines are created with the Automatic Meshing algorithm. The actual mapping of objects onto the mesh is not covered here.

What it does

- Automatic setup of graded mesh in the cartesian x-y-z directions
- All directions or a subset of the cartesian directions can be automatically meshed. The other directions keep their (e.g. manually setup) mesh.
- Automatic meshing is triggered
  - By the user **Create Mesh**
  - Before a simulation is started, if meshing mode is not set to "manual" in the automesh setup.

What does it depend of?

- Automesh settings e.g. Mesh resolution (coarse, fine, medium etc.)
- Simulation settings e.g.
  - highest simulation frequency
  - simulation target frequency
  - boundary conditions, if "add space" is chosen
- Object geometry
- Object properties
- Special "Mesh-hint" property objects

Short algorithmic description

- Preparation
  - The automesh algorithm works for each cartesian direction separately (one by one, starting with the x-direction).
  - First the bounding box of the discretisation e.g. x_{min} and x_{max} is determined
    - either by the structure bounding-box[1], or
    - by user settings (Mesh Setup - Simulation Box)
  - If the automesh-type is neither "user-setup" nor "advanced":
    - A default minimum-mesh-size d_{min} (Target minimum) is calculated depending on the mesh resolution.

$$d_{min} = min\big((x_{max} - x_{min})/intervals, \lambda_{min}/N_\lambda\big)$$

λ_{min} is calculated at the max. frequency (epsr = 1)

Resolution	$intervals$	N_λ
Coarse	1000	10
Medium	2000	15
Fine	4000	20

[1] Objects (e.g. Ports, Library Objects) may include their own hints to calculate their Bounding-Box which may extend the overall bounding box

(d_{min} is also adjusted for group default height resolution.)

* A default absolute-minimum-mesh-size a_{min} is calculated depending on the "Advanced/Relative Drawing Accuracy" option.

  $a_{min} = (x_{max} - x_{min})/accuracy$

* Number of mesh-intervals in objects are set-up
  - basic: number of intervals for basic object hits
  - normal: number of intervals for normal object hits

Resolution	basic	normal
Coarse	1	3
Medium	2	4
Fine:	3	5

- Object Collection
  - Objects are collected by object signature in signature groups. Object signature includes:
    * properties
    * object extrusion height, if any
    * object priority
  - The signature groups are merged and analyzed for
    * Needed mesh planes (blue lines in 2d display): Edge detection
    * Needed mesh density (red lines in 2d display): Interval detection (dependent of mesh-interval settings and object properties)
    * Flat metal processing: signature groups may be identified as "flat" in one direction, those signature groups will be flattened completely and may cause other signature groups' height changes.
- Discretisation Calculation
  - Generate x,y rsp. z discretisation from the detected edges and intervals of all enabled objects.
  - Settings of d_{min} and a_{min} are recognized as well as mesh hints here.
  - In a non-PGA Setup, edges may be moved using the 1/3 - 2/3 rule.

22.2. Manual Meshing

The manual mesh can be set up in the **2D Design** mode which displays the **mesh bars** on the right side and bottom.

The manual meshing can be enabled by

- Setting **Simulation Setup - Mesh Tab - General - Mode** to **Manual**
- Pressing the **left** mouse button on one of the mesh bars and confirming dialog boxes

Mesh action buttons are available after selecting one of the mesh bars.

- **Select** mesh bar: Left click on bar (light green color)
- **Deselect mesh bar**: Click **Esc** key or button



Figure 22.5.: Selected Mesh bar

22.2.1. Marking intervals

If mesh lines are already available intervals can be marked with a red dot by:

- left click between lines
- left click and drag over range of lines
- create a 1D arrow and click **Mark Range**
- click **Mark All**



Figure 22.6.: Mesh bar with marked intervals

Marks can be deleted by using **Delete Marks**

22.2.2. Generic discretization

Press **left** mouse button on one of the mesh bars:

- **Generic Discretization** Deletes initial mesh and inserts fixed lines on object edges
- **Smooth Discretization** Deletes initial mesh and inserts a graded mesh between every pair of fixed lines
- **Smooth Discretization Marked** In marked intervals inserts a graded mesh between a pair of fixed lines

22.2.3. Delete Lines

Press **left** mouse button on one of the mesh bars:

- Press the **middle** mouse button on one line in the mesh bar
- Press and drag the **middle** mouse button over the mesh bar
- **Delete Mesh** Delete all mesh lines
- **Delete Mesh Filling** Delete only un-fixed (red) mesh lines
- **Delete Fixed Range** Delete range defined by entered arrow
- **Delete Range** Delete range (only un-fixed lines) defined by arrow

22.2.4. Add lines

Press **left** mouse button on one of the mesh bars:

- Press the **middle** mouse button on empty space of the mesh bar to set a fixed line
- **Insert Symmetric**, enter δ: Add mesh lines in the marked intervals with the spacing of δ
- **Subdivide**, enter n: Add mesh lines in the marked intervals with n equal subdivisions
- **Add Fixed Lines** Add fixed lines defined by points or tips or arrows
- **Add Unfixed Lines** Add un-fixed lines defined by points or tips of arrows

22.2.5. Edit Lines

Press **left** mouse button on one of the mesh bars:

- **Fix Range** Change un-fixed to fixed lines in the range defined by a 1D arrow
- **Unfix Range** Change fixed to un-fixed lines in the range defined by a 1D arrow
- **Copy Marked** Copy marked mesh lines to position defined by a 1D arrow
- **Shift Marked** Shift marked mesh lines to position defined by a 1D arrow

22.2.6. Port discretization

- Select Port, select mesh bar Add mesh proposed by the port
- Select Port, select mesh bar Add local mesh proposed by the port

23. Simulation Setup

When EMPIRE XPU™ is started, some global simulation parameters have already been entered or set by an opened project. To access and adjust these global parameters press the button **Simulation Setup** .

Buttons at the bottom are available which apply to all tabs:

- **Reset** Apply default settings
- **OK** Use current settings and close window
- **Advanced** Show more settings if enabled
- **Import** Load settings from a file
- **Export** Save current settings in a file
- **Cancel** Disregard current settings and close window
- **Help** Open the manual at the respective help page

23.1. General Setup

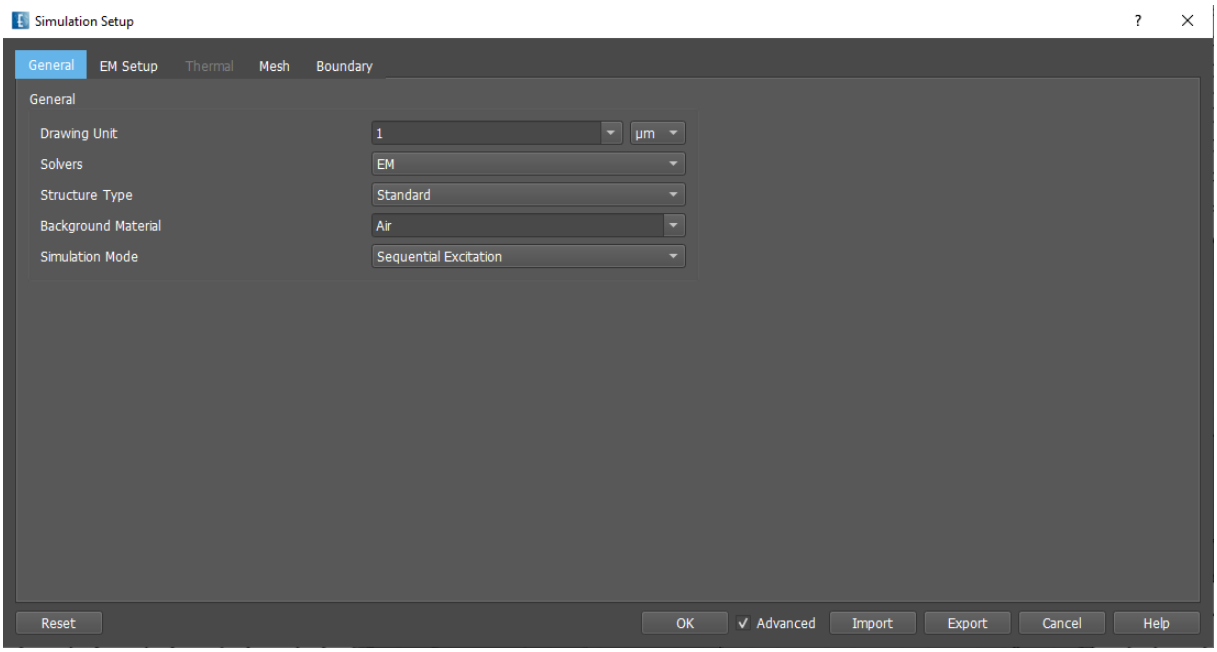


Figure 23.1.: Global simulation parameters

- **Drawing Unit** The drawing unit determines the relation between one unit in the drawing plane and the geometry in reality. The default is $1\mu m$ and can be changed according to the entries in the list. If the unit is changed the user is asked if existing objects should be scaled to the target unit. Note, that only geometry values are changed and does not affect other entries, such as variables, mesh hints, boundary distances, etc.
- **Solvers**
  - **EM** EM Simulation only
  - **EM+THERMAL** EM and subsequent thermal simulation
  - **THERMAL** Thermal Simulation only
- **Structure Type** Determines distance to boundary, number of time steps, energy decay and resonance estimation order depending on typical simulation problems:
  - **Standard**, fast energy decay, low order estimation, e.g. broad band transition
  - **Medium Q System**, medium energy decay, medium order estimation, e.g. narrow band transition
  - **Inductor** $n < 5$, medium energy decay, low order estimation, e.g. single turn inductor
  - **Inductor** $n < 10$, slow energy decay, low order estimation, e.g. few turn inductor
  - **Inductor** $n < 20$, very slow energy decay, low order estimation, e.g. multiple turn inductor
  - **Filter HO** slow energy decay, high order estimation, e.g. high order filter
  - **Filter LO** medium energy decay, medium order estimation, e.g. low order filter
  - **Antenna** same as **Standard** but adds space to open boundary
  - **Antenna efficiency** Here, the number of timesteps is increased and the oversampling factor is enlarged to ensure an accurate determination of port power and radiated power.
  - **Waveguide Structure**, medium energy decay, high order estimation, center frequency above cut-off frequency
  - **Waveguide Filter**, slow energy decay, high order estimation, center frequency above cut-off frequency
  - **Waveguide Antenna**, fast energy decay, low order estimation, center frequency above cut-off frequency
  - **Antenna Field Source**, fast energy decay, zero order estimation, modulated pulse at target frequency with 30% bandwidth
  - **User Setup (Advanced)** Here, the end criteria (last timestep and energy decrement) and resonance estimation order have to be defined by the user.
- **Background Material** Material which is used to fill the space between objects and boundaries.
  - **Air** Default material
  - **Vacuum** permittivity = 1
  - **PEC** Ideal conductor, e.g. used for waveguide networks
  - **epsr=val** Can be used if surrounding matter is different to vacuum, e.g. soil, water
- **Simulation Mode** This mode determines how multiple ports are excited and which results will be created
  - **Sequential Excitation** Subsequent excitation and simulation of all ports marked with "E". Each result folder contains one column of the S-Matrix

- **Sequential Excitation + Y-Parameter Exraction** Subsequent excitation and simulation of all ports marked with "E". Calculation of S-Matrix from Y-Parameters and generation of Touchstone file with Y_Parameter processing (ypar folder). This mode is used for circuit simulation as part of the postprocessing. If selected, the user can choose if conventional S-parameter calculation should be enabled or not.
- **Simultaneous excitation (Active Impedance)** Simultaneous excitation and simulation of all ports marked with "E". Result folder contains diagonal elements of S-Matrix. If selected the user can choose the power normalization for transmission values.
- **Time Domain Reflectometry** Sequential excitation and simulation of all ports marked with "E". If selected the user can modify TDR specific values in the **EM Setup** Tab.

As an example for the **Structure Type** setting the following table is obtained for the standard MSL template:

Structure Type	Timestep Limit	Energy Decay	Estimation Order	Additional Space	Modulated Pulse
Standard	20k	40 dB	40	no	no
Medium Q	100k	60 dB	100	no	no
High Q	700k	70 dB	200	no	no
Inductor 5	350k	50 dB	30	yes	no
Inductor 10	550k	50 dB	30	yes	no
Inductor 20	800k	50 dB	30	yes	no
Filter High Order	200k	60 dB	200	no	no
Filter Low Order	100k	60 dB	100	no	no
Antenna	80k	40 dB	40	yes	no
Antenna Efficiency	300k	60 dB	100	yes	no
Waveguide	450k	60 dB	150	no	yes
Waveguide Filter	950k	60 dB	400	no	yes
Waveguide Antenna	100k	40 dB	100	yes	yes
Antenna Field Source	45k	30 dB	0	yes	yes

23.2. EM Setup

23.2.1. Frequency

The values entered here define the range of the frequency domain results. The End Frequency is also used as input for automatic meshing. If possible the simulation will be carried out for a larger frequency range, including DC to shorten the pulse (depending on largest cell size).

- **Start Frequency** Used for DFT
- **End Frequency** Used for DFT and maximum cell size criterion
- **Number of Frequency Points** Support points used for DFT of time domain signals
- **Target Frequency** Used as center frequency in QTEM ports, modulations, narrow band loss models and default frequency in field dumps

Hint: Pulse width

It can be advantageous to excite with a pulse as short as possible (high bandwidth) to minimize simulation time.

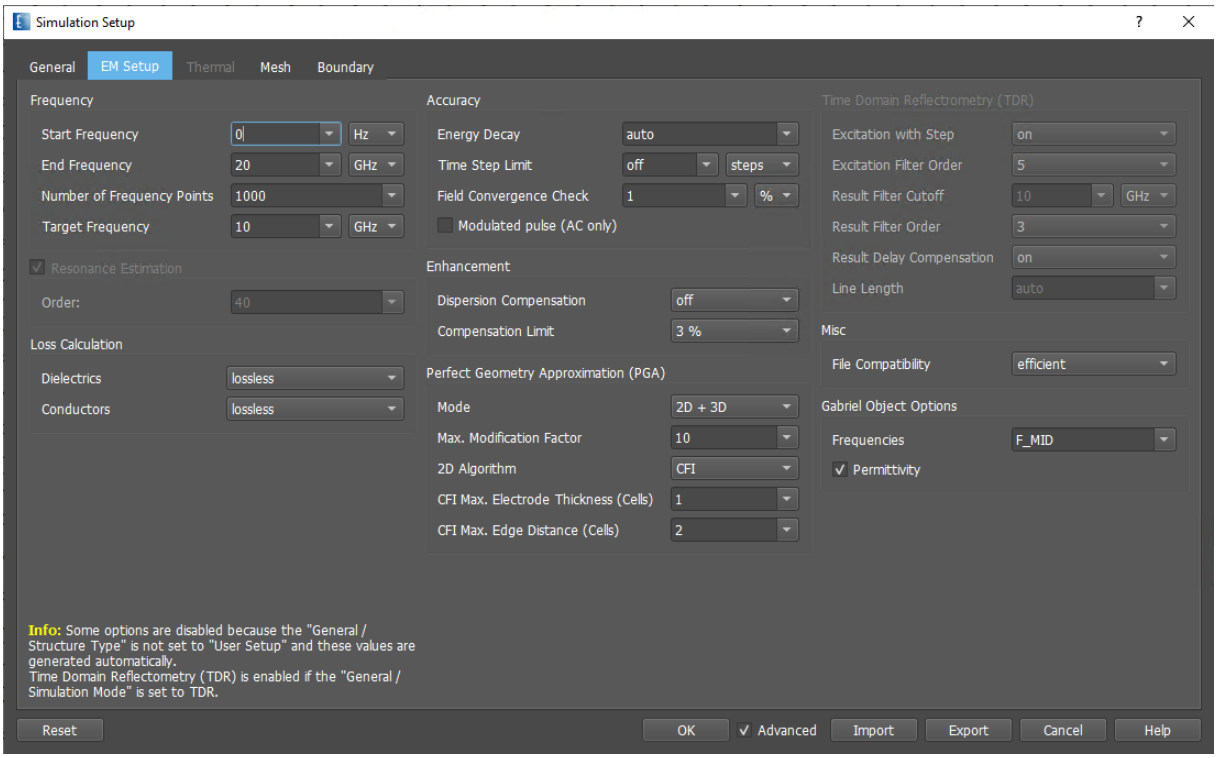


Figure 23.2.: EM simulation parameters

Hint: Pulse shape

If another pulse shape than a Gaussian pulse should be used, it can be set in **Simulation Tab - Excitation**

23.2.2. Resonance Estimation

Accessible if **Advanced** is checkmarked at the bottom

The estimation algorithm is part of the DFT postprocessing. It analyzes the tail of time signals and extrapolates the signal up to infinity. The value may be changed after a simulation to perform a new postprocessing.

- **Order** Number of sampling points of the tail of time signals

Short time signals can only be estimated with low orders. In case of multiple resonances the order should be increased (which requires long time signals).

Hint: Application Note

More information is available in Chapter 29.2.6

23.2.3. Loss Calculation

By default **Dielectrics** and **Conductors** are treated as lossless. This reduces computational effort (less operations and memory consumption) and simulation time for low-Q structures. Reliable results for parameters such as

reflection or coupling are still achieved. In case of high-Q structures (e.g. resonators) or if the transmission losses play a significant role the losses can be included in the simulation at different levels.

Dielectrics

For Dielectrics the **Loss Tangent** should be constant over frequency. In order to model losses, the FDTD algorithm only supports constant conductance and higher order time derivatives (Debye material) to model frequency dependent dielectrics.

For narrow band losses an equivalent conductance value is used for the target frequency. For medium and broadband losses a first or second order Debye model is applied.

- **lossless**, ideal dielectrics
- **narrow band (target-freq)**, equivalent conductivity $\sigma = 2\pi f \varepsilon_0 \varepsilon_r \tan\delta$
- **medium band**, 1st order Debye model of constant loss tangent, optimized for lower frequency range
- **broad band**, 2nd order Debye model of constant loss tangent, optimized for lower frequency range
- **medium band (target-freq)**, 1st order Debye model of constant loss tangent, optimized for target frequency
- **broad band (target-freq)**, 2nd order Debye model of constant loss tangent, optimized for target frequency

Conductors

Conductors can be accurately modelled with a bulk loss model having a constant conductivity. This model is recommended if

- the skin depth is larger or in the range of the metal thickness
- the field penetration through a thin conductor should be evaluated

But this requires to resolve the skin depth with many cells and leads to long simulation times.

To avoid this, special surface loss models have been developed and can be used in EMPIRE XPU™ which do not require to resolve the skin depth so that planar metallization can be treated as flat and opaque. If the metal is resolved nevertheless the field inside is zero. These loss models are generated for the entered frequency range for broadband and medium band. Here, the simulation effort is enlarged so that memory increases and performance drops.

A narrow band loss model can be applied, too with no impact on performance, which calculates the respective resistivity **Rsquare** at a certain frequency. In this case the target frequency has to be entered.

- **lossless**, ideal conductors (PEC, infinite conductivity)
- **narrow band lossy**, constant conductivity, typically ±10% bandwidth
- **medium band lossy**, 1st order approximation of skin effect typically ±50% bandwidth
- **broad band lossy**, 2nd order approximation of skin effect, typically ±95% bandwidth
- **bulk loss model**, should be applied in case the conductor thickness is smaller than the skin depth

23.2.4. Accuracy

- **Energy Decay** End criterion for the simulation. The value specifies the energy level below maximum. If set to **auto** the number will be derived from the setting in the **Structure Type**.
- **Time Step Limit** End criterion for the simulation. The value specifies the maximum number of time steps. If set to **auto** the number will be derived from the setting in the **Structure Type**.
- **Field Convergence Check** If enabled, the field convergence of a field dump is evaluated during the simulation and printed in the log window
- **Modulated Pulse (AC only)** If enabled, the Gaussian pulse will be modulated with the target frequency.

23.2.5. Enhancement

A directive correction for numerical dispersion can be applied for accuracy enhancement. If needed this global setting can be switched off for groups and objects by applying property **No Dispersion Correction**

- **Dispersion Compensation** Specify direction for dispersion correction
- **Compensation Limit** Specify accuracy for dispersion correction

Notes

- The dispersion correction is useful for structures where one dimension of the solvable domain is much longer than the other two.
- If the solvable space is large but quasi-cubical, then this correction cannot really help, since it is only applied in one direction.
- Useful, e.g for sPD calculations where the numerical dispersion messes up the phases of the forward and backward travelling waves
- To correct this, the speed of light is applied in the direction of propagation instead of the diagonal of a cell. The correction is applied to the FDTD coefficients directly.
- The correction does not alter the mesh, it alters space itself.
- This correction makes dispersion worse in the other two directions. This can damage the behaviour of resonant structures, e.g., antennas and filters. Therefore, the volume around such structures needs to be excluded from the correction by applying a special property to the Groups.
- If the mesh is fine enough (like lambda/40), then no correction is needed.
- The compensation limit can be left at the default value of 3%. The actual value is reported in fdtd.log.

23.2.6. Perfect Geometry Approximation (PGA)

This algorithm recalculates FDTD coefficients for conductors. It is based on a local static field estimation for off-grid, rounded and flat metal objects. It reduces the staircasing effect while the FDTD performance (cells/sec) is maintained but decreases the time step size by the square root of the modification factor.

- **Mode** Can be limited to 2D or 3D metal objects, or switched off
- **Max. Modification Factor** for FDTD coefficients. Big values indicate large corrections but may decrease the time step size.
- **2D Algorithm** 2D coefficients determined by conformal mapping (CFI) or local field estimation (PGA)

- **CFI Max. Electrode Thickness** Apply algorithm to areas with metal thickness less or equal number of cells
- **CFI Max Edge Distance** Apply algorithm to areas with edge distances less or equal number of cells

23.2.7. Time Domain Reflectometry (TDR)

TDR can be used to verify impedance calculations, detect abrupt impedance changes, and optimize transmission line (TL) discontinuities and transitions between different types of TL. An application note including examples is available. TDR parameters can only be accessed if the **Simulation Mode** in **General** is set to **Time Domain Reflectometry (TDR)**

- **Excitation with Step** If on, excitation is carried out with the unit step. If set to off the excitation may be user-defined
- **Excitation Filter Order** Order of the low pass filter used for excitation
- **Result Filter Cutoff** Should be set appr. 1.5 - 2 times target frequency
- **Result Filter Order** Order of the low pass filter used for result evaluation
- **Result Delay Compensation** Undo the time shift in the results caused by the filter
- **Line Length** Length in drawing units for automatic determination of a time step limit

Hint: Application Note

More information is available in Chapter 29.2.21

23.2.8. Misc

- **File Compatibility** If set to efficient, the preprocessing is accelerated for non-solid geometries. To be set to compatible, if older kernel version (< 8.2) should be used

23.2.9. Gabriel Object Options

Treatment of Gabriel material for simulation. A Gabriel material (tissue) can only be modeled for a single frequency which can be set here. For sequential multi-port excitations a list of frequencies may be defined. For very low frequencies, permittivities tend to infinite but can be optionally ignored because conductivities are the dominant effects.

- **Frequencies** Default is the **Target Frequency** (F_MID).
- **Permittivity** If disabled, a relative permittivity of vacuum is used while the conductivity is kept.

23.3. Thermal

Only available if **Solver** is set to **Thermal**

Hint: Application Note

More information is available in Chapter 29.2.22

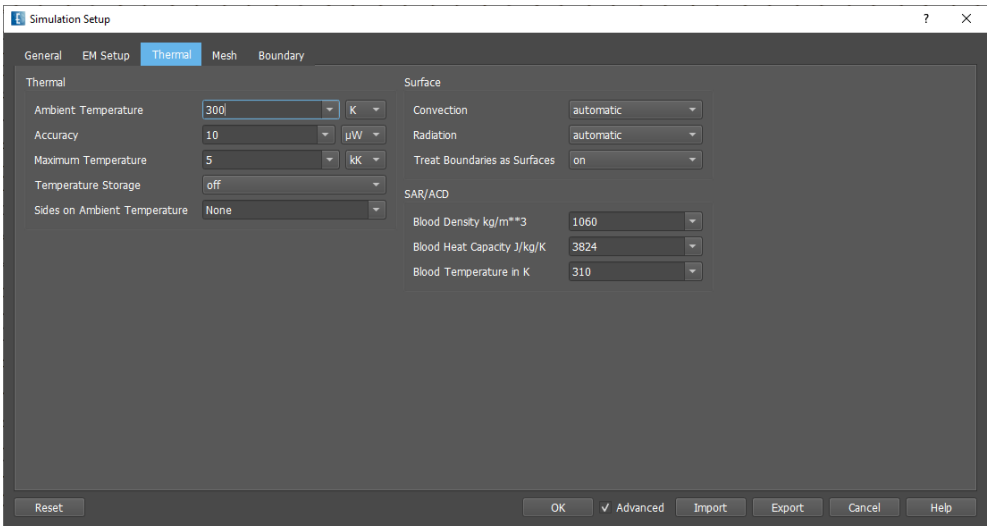


Figure 23.3.: Thermal simulation parameters

23.3.1. Thermal

- **Ambient Temperature** Defines the temperature of the surrounding air and bounding box side if applicable
- **Accuracy** Defines the acceptable error for the solution convergence and the iteration end criterion
- **Maximum Temperature** Upper limit for the possible temperature
- **Temperature Storage** Number precision, can be either 8bit or 16bit
- **Sides on Ambient Temperature** Defines a heatsink with ambient temperature at boundaries of the structure

23.3.2. Surface

By default the surfaces for convection and radiation to the surrounding (air, zero thermal conductivity) are determined automatically if **Treat Boundaries as Surfaces** is set to **on**. For special purposes they can be set manually in the object properties or switched off.

- **Convection** Heat transfer from surface to the surrounding using a convection model. The corresponding **Surface Heatsink coefficient** has to be set in the object property
- **Radiation** Heat transfer from surface to the surrounding using a radiation model. The corresponding **Surface Radiation Emission Coefficient** has to be set in the object property
- **Treat Boundaries as Surfaces** Can be switched off, e.g. in case of truncated structures. Applies to both convection and radiation.

23.3.3. SAR/ACD

The following parameters are only used for thermal simulations invoking body models. The tissue thermal parameters are defined in the voxel files. Blood perfusion plays a major role for heat transfer in body models.

- **Blood density** Specific density of the blood
- **Blood Heat Capacity** Specific Heat Capacity of blood

- **Blood Temperature** in Kelvin (310K = 37°C = 98°F)

An additional body heat transfer mechanism can be set as **Basal Metabolism** in the respective object property as a thermal parameter.

23.4. Mesh

By default the automatic meshing is enabled and creates a suitable mesh for the objects and sources and enterd simulation parameters (e.g. end frequency, structure type, boundaries).

The settings of the automatic meshing can be adjusted in the **Simulation Setup** in the **Mesh** Tab.

Detailed information about Meshing can be found in Chapter 22.1

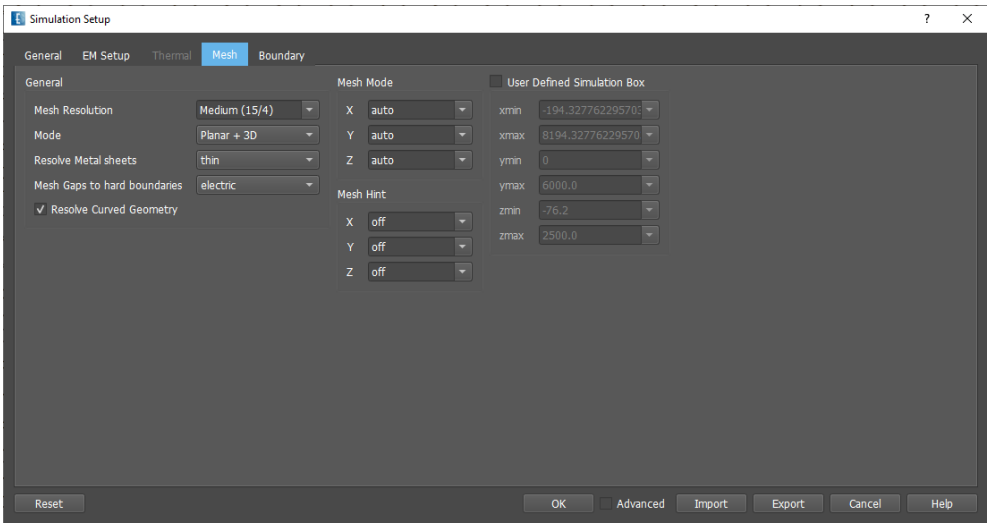


Figure 23.4.: General Mesh Adjustments

Hint: Application Note

More information is available in Chapter 29.2.13

23.5. Boundary

23.5.1. Boundary conditions

These parameters determine which boundary conditions will be applied at the six sides of the simulation domain.

- **Electric** Tangential electric field is forced to be zero at the outermost grid line. Used for (lossless) ground planes, metal packages, or symmetry planes
- **Magnetic** Tangential magnetic field is forced to be zero in the middle of the outermost cell. Used to truncate simulation domain if radiation can be neglected at the boundaries. Further, it can be used as symmetry plane.

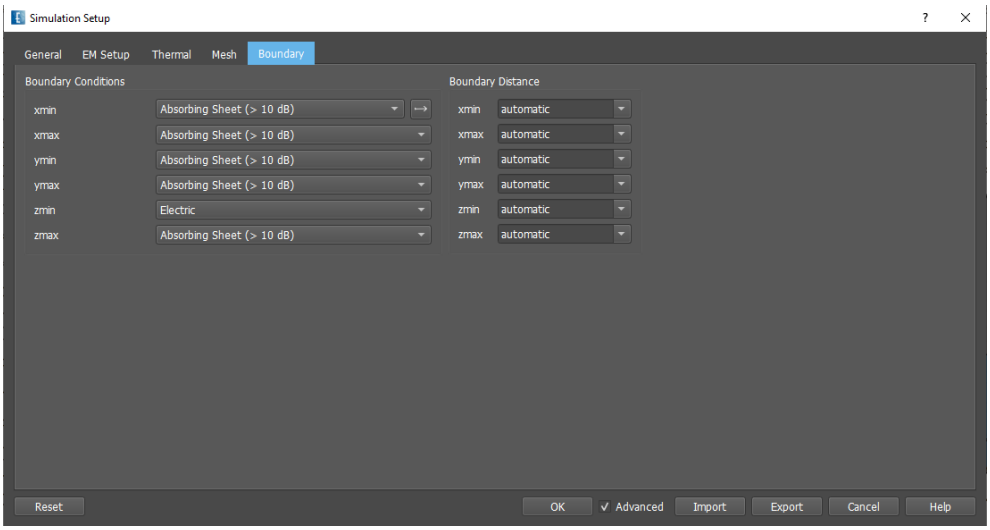


Figure 23.5.: Boundary setup

- **Absorbing sheet** A sheet with (default) 377Ω free space impedance is placed at the boundary which absorbs radiation directed normal to the boundary. Used for high directivity radiation or to suppress box mode resonances in simulations with low radiation.
  - **add. space** Additional space can be added and if needed.
  - **Z_ref** the free space impedance can be adjusted if needed.
- **Absorbing n** Radiating boundary condition with **n** Perfectly Matched Layers (PML). The higher the number of layers the lesser reflections will be produced at the boundary (but slows down simulation speed). The layers are placed outside the simulation domain.
  - **add. space** Radiating boundary condition with additional space to the structure.

A button is available in row **xmin** ⟷ as a shortcut to apply the boundary condition to all sides.

The PML is placed beyond the boundaries. So the computation domain includes simulation box and PML area. The size of the PML is determined by the following

- There will be N cells beyond the boundary. The number N can be set by the user in the boundary conditions. The default number is 6 (cells)
- The cell size in wall direction is determined by the outermost cell size

23.5.2. Boundary Distance

Settings as explained in chapter 22.1.6

23.6. Excitation

More simulation parameters can be set in the **Simulation** Tab by clicking the button **Simulation Setup** after clicking **Pre-processing**. Some parameters (Bandwidth, Center frequency, Maximum Number of timesteps, Energy, ...) can only be set if item **Structure Type** is set to **User Setup**.

Here, the exciting pulse and the end criteria can be adjusted.

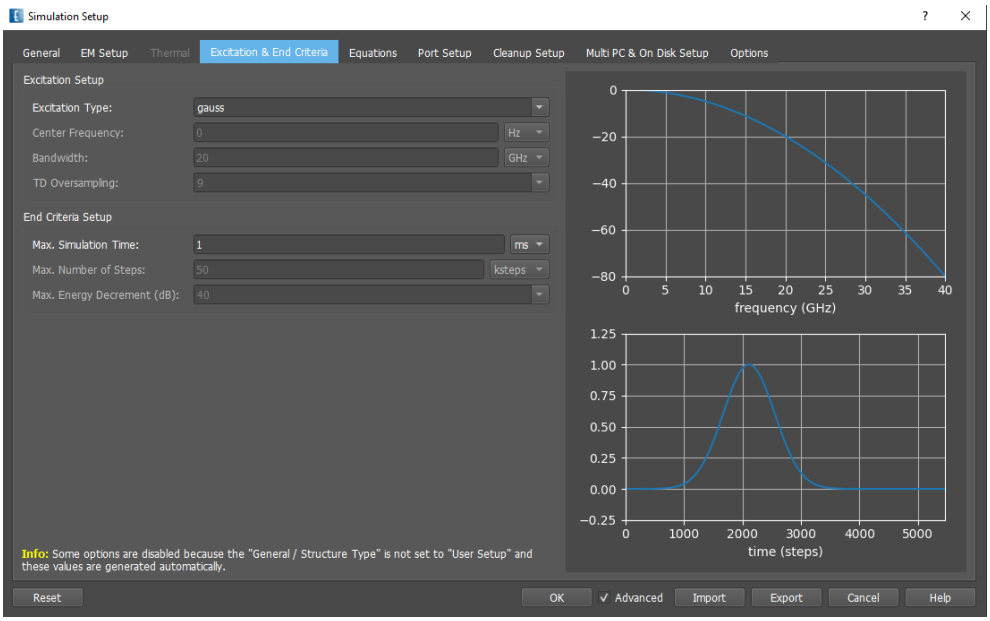


Figure 23.6.: Excitation setup

23.6.1. Excitation Setup

- **Excitation Type** Shape of pulse
- **Center Frequency** Modulation frequency
- **Bandwidth** Width of Pulse
- **TD Oversampling** Determines number of time sample points in time domain results

23.6.2. Pulse shape

If the pulse type is set to **gauss**, the center frequency f_0 and the bandwidth $2f_{hbw}$ determine the pulse length according to the following relations:

$$e(t) = e^{-(\frac{t-t_0}{\tau})^2} sin(2\pi f_0(t-t_0))$$

where

$$\tau = \frac{\sqrt{\ln 10}}{\pi f_{hbw}}$$

and

$$t_0 = \frac{\ln 10\sqrt{5}}{\pi f_{hbw}}$$

By default, the bandwidth is derived from the largest cell size and the center frequency is set to 0 to achieve the shortest possible pulse length. Only in case of hollow waveguide excitation or if the "AC only simulation" is enabled the center frequency is chosen differently.

Alternatively, a set of templates is available to be selected or edited to excite with other pulse shapes. Here, the shape is defined by mathematical expressions and time values in seconds.

Pulse shape in Frequency domain

According to the entered values on the left and selected pulse shape, the energy distribution of the pulse in frequency domain is depicted in the top diagram. The maximum energy for a Gaussian pulse is at the center frequency. The curve refers to the result file **ef**.

Pulse shape in Time domain

The preview of the pulse shape refers to the exciting function **et** and is shown in the bottom window on the right where the time is scaled in number of time steps. The time step size is fixed by the discretized structure and written above. On the left, the sampling factor can be adjusted. For the DFT, a minimum of 3 is recommended. This value can be enlarged to obtain smoother time signals.

Hint: Minimum number of time steps

In the pulse preview window, the required number of time steps can be estimated. At least twice the duration given in steps have to be simulated.

23.7. End Criteria

By default the simulation is terminated if the following criterias are fulfilled:

- **Maximum Timsteps** If this number is reached the simulation will be stopped. The number can be changed during a running simulation. A restart is not possible.

OR

- **Convergence criteria:**
  - **Energy Convergence** This criterion is evaluated in time domain and compares the input energy with the remaining amount in the simulation domain

  AND

  - **Field Monitor Convergence** This criterion is evaluated in frequency domain and applies if a field monitor is defined (not available for Multi-PC algorithm)

The values for these criterias can be set in **Accuracy** of the **EM Setup**, see section 23.2.4.

23.7.1. End Criteria Setup

Here are some adjustments to terminate the simulation run if the **Structure Type** is set to **User Setup** in **General** Tab.

- **Max. Simulation Time**: The simulation time is determined by the number of time steps multiplied by the time step size. This limit is useful if simulations with different resolutions should be compared. The default value is set to a high value to ensure that the 2 criteria below will apply in most cases.
- **Max. Number of Steps**: The simulation will terminate if this value is exceeded. This number is determined by the structure size and **Structure Type** parameter.
- **Max. Energy Decrement (dB)**: If the energy left inside the structure is declined to this value, the simulation will be terminated. The default value is determined by the **Structure Type** parameter if not user defined.

Remarks:

- The value of the maximum simulation time is not known in general but depends on the excitation response. Excitation responses for different Structure Types can be set in the Simulation Setup
- If only the maximum energy estimation decrement should be used to terminate the simulation run the maximum number of time steps should be set to a very large value.
- If field monitors are defined the energy end criteria will only be used if the field convergence in 23.2.4 has been achieved.

Hint: Restart

Restarting the simulation after termination is not possible. But it is possible to change these numbers during the simulation.

23.8. Equations

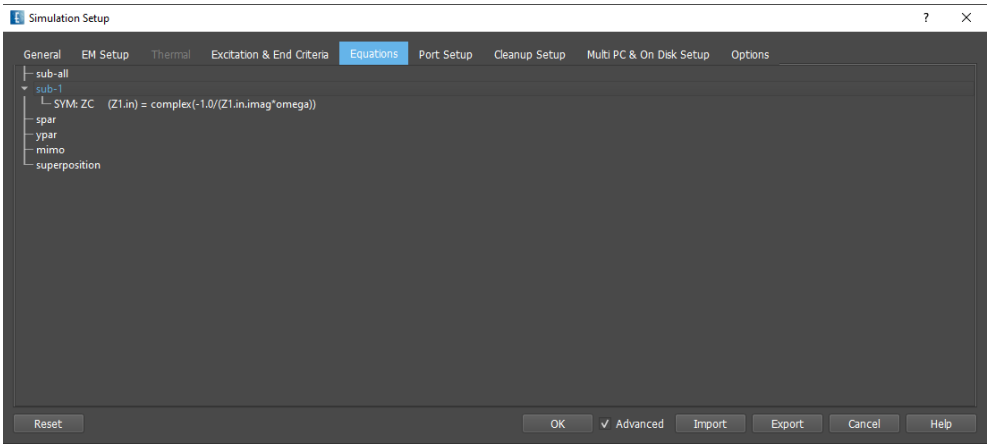


Figure 23.7.: Postprocessing Equation Setup

To obtain more advanced results already calculated data can be further processed.

List of Subfolders

In the left area, the result folders are displayed. An equation can be added with a right click on the respective folder

sub-all Result folder for all sub directories

sub-i (**i** is a number of the port which has been excited)

spar Result folder for all S-parameters

ypar Result folder for all Y-parameters

mimo Result folder for envelope correlation coefficient calculation

superposition Result folder for field superpositions

The button **Delete** applies to the **USERDEF** folders only.

Equations

Equations can be defined by right click on a result folder and select **Add Equation**

23.8.1. Symbolic Equation

Equation for frequency domain files to be composed using an editor or templates.

$y(f) = y_1(x_1(f)) + y_2(x_2(f)) + \ldots$

Usage

1. Right click on the desired subfolder
2. Select **Symbolic equation**
3. Enter destination file name
4. Right click in **Sources** select **Add Source** and file x_1
5. Optionally, add more source files
6. Enter mathematical expression **Term**. A list of **Templates** is available below
7. Optionally define a **Label** to be displayed in the plot

A collection of frequently used equations can be found in the **Template**.

Supported functions

Functions defined in Python libraries **math** and **cmath** are supported. In addition the following abbreviations are added:

- **conj, conjugate** conjugate
- **sign, sgn** result multiplied with sign of independent variable
- **j** complex(0,1.0)
- **f** independent variable
- **x** independent variable
- **omega** 2*math.pi* f
- . dots in source files names (e.g. Z1.in) are replaced by _
- **.diff** suffix for derivative of source files

23.8.2. Calculation

A collection of postprocessing programs which analyze frequency domain result files and create log files.

Currently available calculations:

- **find_dips** Search for minima in the file and write resonance frequencies, bandwidth and quality factor into the log file. The frequency range and the number of minima can be limited.

The entries in the log files may be used in optimization goals and can be plotted in the **Sweeps** Tab.

23.8.3. File Copy

Used to copy result files from other result folders (e.g. from sub-1 to sub-2).

$y(t, f) = folderexpression/x(t, f)$

Usage

1. Right click on the desired subfolder
2. Select **File Copy**
3. Enter destination file name
4. Search source file or enter copy expression, e.g. $ut3 = ../sub - 3/ut1$

23.8.4. Touchstone File

Generates the S-Matrix in Touchstone format. n ports have to be excited if an $n \times n$ Matrix is expected. Available for **spar** and **ypar** folders. In latter case, the S-Parameters are derived from the Y-Matrix.

$file.snp = Touchstone(n \times n$ -S-Matrix)

Usage

1. Right click on the desired subfolder
2. Select **Touchstone File**
3. Enter **Destination Filename**
4. Select **S Parameter Ports**

23.8.5. Field Superposition

Both nearfield and farfield results can be superimposed here by defining equations. This can be useful for circular polarized results if the two polarizations have been simulated subsequently.

Usage

1. Right click on the **superposition** folder
2. Select **Field Superposition**
3. Enter **Destination Filename**
   - (non Legacy): Select source files to be superimposed and respective norms (may be complex, e.g. ef=j)
   - (Legacy): Select source files with names (e.g. A and B) to be superimposed and define equation in **Terms** (e.g. A+j*B)

Files have to be equal in size. For compatibility reasons this option is availble for legacy files, too.

23.8.6. Far Field Calculation

This setup is available for the far field calculation of a superimposed Huygens surface (recording of the far field monitor)

Usage

1. Right click on the **superposition** folder
2. Select **Far Field Calculation**
3. Setup far field processing, similar to Ch. 19.3.2
4. In **Advanced** enter **Near Field File** (Huygens surface recording, e.g. Farfield_1) as defined in **Field Superposition** equation

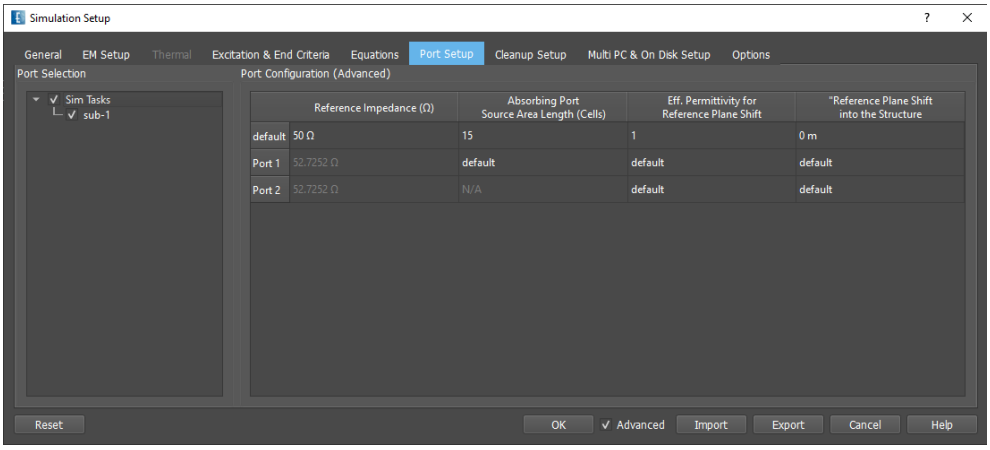


Figure 23.8.: Port setup

23.9. Port Setup

Port Selection

Here, port excitations can be pre-selected / deselected for simulation. The settings are only applied if no pre-processing is executed yet.

Port Configuration (Advanced)

- **Reference Impedance**: This value is used for the separation of incident and reflected waves, e.g. **ut1.inc, ut1.ref**, in the case of concentrated port excitation and for Y-Parameter calculations. It can be only adjusted if the **Internal Port Impedance** is not used as **Reference Impedance** (Rcopy=Off in tn respective **Port Parameters.**
- **Absorbing Port: Source Area Length (Cells)**:
  1. $N = 3$ for waveguide excitation with TE- or TM-modes
  2. $N > 2D_f/\delta x_b$ for matched source excitation, where D_f is the effective field diameter and δx_b is the first discretization step at the excitation boundary.
- **Effective Permittivity**: Has to be entered if a reference plane shift should be applied in the postprocessing. For the feed lines in the port library, an **Info** window displays this value depending on the line parameters.
- **Reference plane Shift into the structure (S-Parameters only)**: The port reference plane can be adjusted if needed. This acts on S-Parameters during post processing (Value may be negative)

23.10. Multi PC / On-disc Simulation Setup

The setup defines the hosts (Slave PCs) which are used for parallel computing. The Master PC can be local or remote if defined in the **Host Setup**. Further the hard disc support can be enabled for single PC simulations.

- **Simulation Mode** Can be set to **Single PC** (default), **On Disk** which extends the RAM or **Multi PC** for parallel execution of a single job.

- **RAM Limit in MB** This number (or a set of numbers) can be entered which determine the memory size in MB for RAM allocation on the hard disk.
- **Number of slaves** The number should be equal to the number of hosts selected in the host list on the right.
- **Save virtual memory** If enabled, field dumps will be swapped to disc to save virtual memory.
- **Algorithm** Depending on the net speed the algorithm can be changed from **block** to **row**.
- **Communication** Specifies the protocol which is used to communicate between hosts
- **No Harddisc Reclaim** Can be disabled in case of connection problems

Hint: Application Note

More information is available in Chapter 29.2.15

23.10.1. Slave Host Setup

Here, the slaves for multi-PC simulation can be selected if the EMPIRE XPU™ **Legacy Server** is running on the hosts. The hosts are detected automatically (host broadcast) if the hostnames are known in the network (DNS, host table). They can also be added manually by entering the hostname in the field "Host" and pressing **Add**

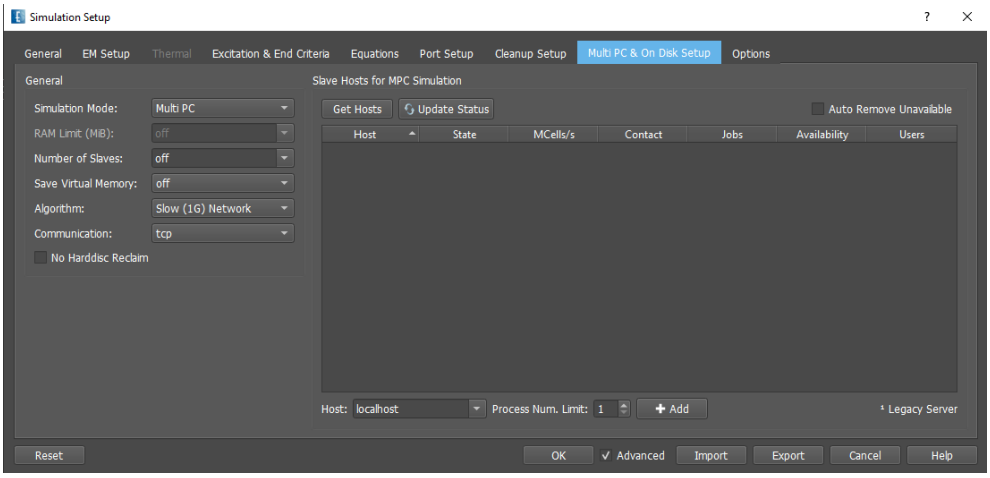


Figure 23.9.: Multi PC Solver setup

23.11. Options

Admin Switches

- **Wait for licenses**: If more remote hosts are enabled than licenses available in the network, this switch causes simulation tasks to wait until licenses are checked out again.
- **Number of Threads**: Number of cores on a single PC used for simulation. In case of NUMA CPUs the number of threads are automatically set to a multiple of NUMA nodes.
- **Simulator job priority**: Sets the priority of a simulation job. Can be verified with the Task Manager.

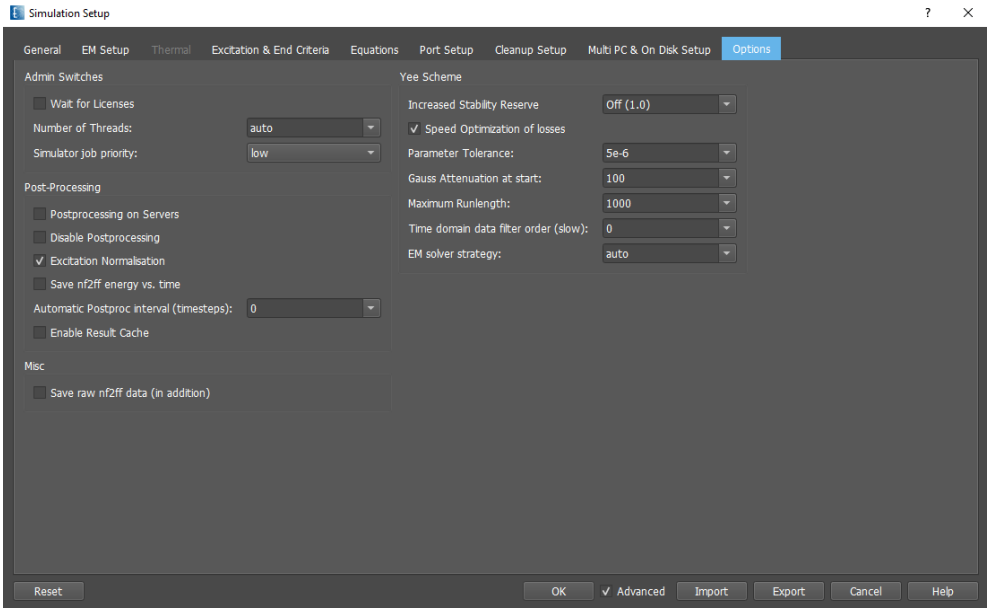


Figure 23.10.: Simulation Options

Post Processing

- **Postprocessing on Servers**: If enabled, the post processing for far field calculation or SAR precalculation will be carried out on the remote servers
- **Disable Postprocessing** Can be used e.g. if only time signals are required.
- **Excitation normalization**: By default, the energy distribution of the incident pulse over the frequency is compensated by the normalization function **ef**. Can be switched off if absolute values are required over frequency.
- **Save nf2ff energy vs. time**: Can be enabled for convergence studies of the far field results
- **Automatic Postproc interval (minutes)**: Generates field dumps and far field recordings during the simulation, e.g. to check the time convergence or for backup purposes.
- **Enable Result Cache**: Used for 2D Result Data

Misc

- **Save raw nf2ff data (in addition)**: If enabled, additional data is saved in boundary nearfields for farfield transformation (e.g. Farfield_1 file) for compatibility reasons. If disabled, the memory requirement is reduced by ignoring additional data

Yee Scheme

- **Increased Stability reserve**: Reduces the calculated maximum time step. This can help to avoid long term instabilities. A numerical factor can be set, e.g. 2 will set reduce the timestep size to 50%.
- **Speed Optimization of losses**: Can be disabled to be compatible with old versions
- **Parameter tolerance**: Level, at which relative FDTD coefficients are treated as equal.
- **Gauss Attenuation at start**: Level of incident pulse at time step 0.
- **Maximum Runlength**: Number of Yee cells used in one system call

- **Time domain data filter order**: Can be used to filter the voltage and current time signals. The higher the filter order the slower the simulation performance
- **EM solver strategy** If set to auto, the fastest strategy for multiple time stepping is tested every simulation. Can be set manually here.

24. Simulation

This chapter describes the different simulation and processing options.

Hint: Project folder

A project file is stored with ending .emx. It automatically creates a subfolder with the same name. All simulation data will be stored within this folder path. Use **Save as** to generate copies of the project.

24.1. Preprocessing

After the model has been set up, the input data is converted into a discretized format (file ending **.acad**) to be further processed by the EMPIRE XPU™ kernel. This is done by

- Clicking the button **Start Simulation** in the **2D/3D Design** Tab to generate the **.acad** file and start a basic simulation immediately.
- Clicking the **Preprocessing** button in the **Simulation** Tab to generate the **.acad** file to prepare the simulation
- Clicking the **Start Complete Simulation** button in the **Simulation** Tab to generate the **.acad** file and start the simulation

In order to setup a simulation batch queue (see Ch. 24.6) these **.acad** files have to be available in the project folder.

The preprocessing also creates a logfile **.log** with project name which saves statistics with object and mesh info of the preprocessing.

24.2. Quick Start

A Simulation Control Window is opened by pressing the **Start Simulation** button from one of the Design tabs.

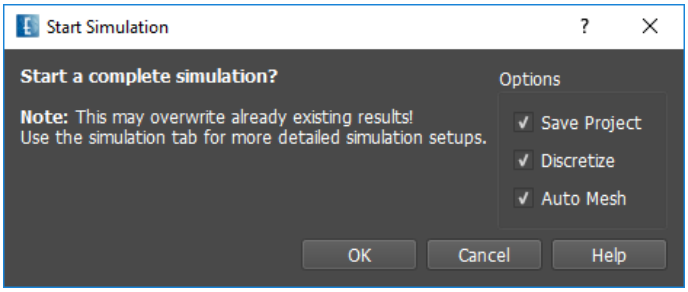


Figure 24.1.: Simulation Start Window

Some options can be selected and de-selected

- **Save Project** Saves the project file if enabled (update project file .emx)
- **Discretize** Maps the structure on the mesh (create new preprocessing file .acad)
- **Auto Mesh** Creates a new mesh before discretization

Simulation statistics, structure info and some result are printed in a log window and written into log files which can be browsed by using the **Utilities - Log Viewer** menu.

24.3. Simulation Control

In the **Simulation** Tab all jobs of the current project are listed on the left side. Each job can be selected in order to show its **Status** window having a **Log** and **Plot** Tab.

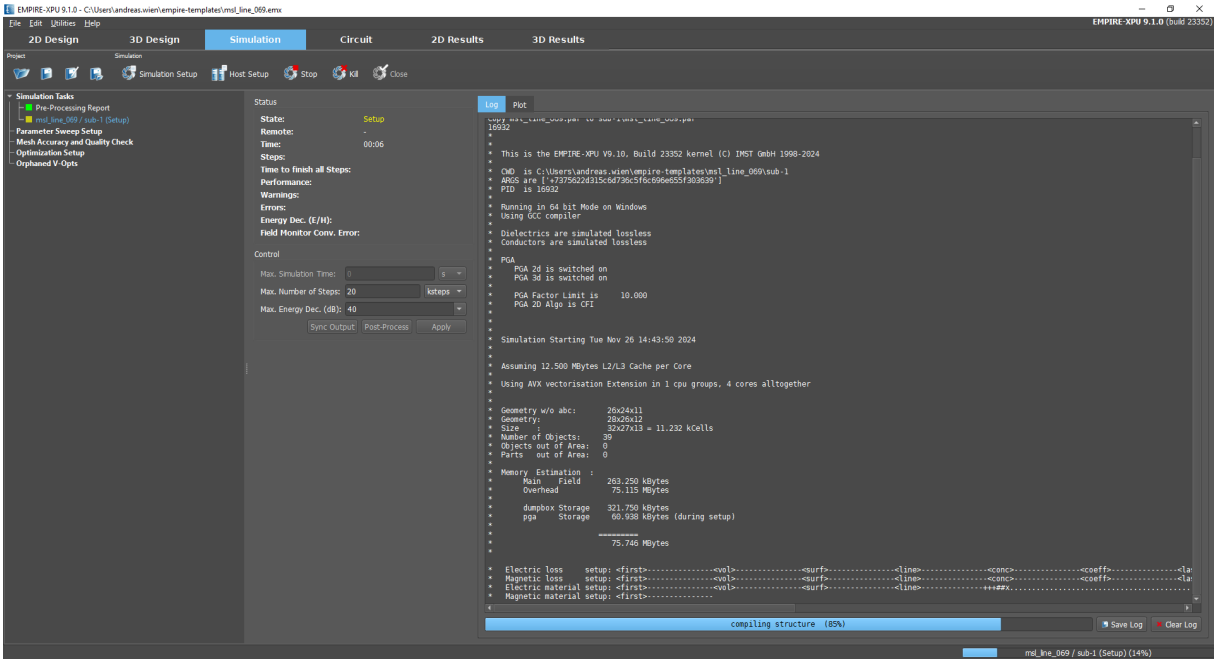


Figure 24.2.: Simulation Log Window

24.3.1. Simulation Log

The content of the window will be stored as **fdtd.log** in each simulation sub folder.

Before a simulation starts the **Log** Tab shows statistics of the pre-processing such as

- **Software Version** of the FDTD kernel
- **Simulation folder and Process ID**, e.g. to be identified with the Windows task manager
- **Operating System and Compiler Version**
- **Loss treatment** which is set in the **Simulation Setup**
- **PGA usage** (Perfect Geometry Approximation algorithm)
- **Start Time** clock footprint

- **Hardware and Features** Detected CPU
- **Number of Cells and Parts** Statistics of discrete model
- **Memory Estimation** of fields, monitors and overhead
- **Compilation Progress** of the on-the-fly code generation
- **Time Step Size** used and (estimated) time step in seconds [1]
- **Sampling** Factor used for storing time signals
- **Threads and XPU algorithm** Used cores and algorithm for multiple time stepping
- **Cache Hits** Efficiency for coefficient re-use
- **Setup time** Time needed for compilation

During the simulation the log window shows:

- **Multistep Strategy** Opimimum tested in the first timesteps
- **Current Time Step and Performance** in $Mcells/s$
- **End Criterion** Reason for simulation end
- **End Time** Clock footprint
- **Simulation time** Time needed for simulation including preprocessing

After simulation the postprocessing is started and listed in the window:

- **Postprocessing** Actions depending on the application (e.g. DFT, Far field transformation, etc.)

24.3.2. Simulation Plot

If the simulation is running the progress is displayed in the **Plot** tab showing the energy convergency over the timesteps. The plot can be changed to

- **Speed Plot** Current performance in $MCells/s$
- **Energy Convergence** E- and H- energy inside the simulation volume with respect to the maximum
- **Field Monitor Convergence** Accuracy of the frequency domain field monitors
- **Voltages** Signals at the ports
- **Voltages (log)** Signals at the ports in logarithmic style

24.3.3. Simulation Actions

The following table summerizes different actions during a running simulation:

Action	Meaning	Simulation	Near Field output	DFT processing e.g. S-Parameters	Far Field processing
Kill	End job immediatly	Terminate	No	No	No
Stop	End job at the earliest	Terminate	Yes	Yes	Yes
Sync Output	Save field records	Continue	Yes	No	No
Post-Process	Create intermediate results	Continue	No	Yes	Yes (after Sync)
Apply	Update End Criteria	Continue	No	No	No

[1] Estimation without PGA or QTEM ports

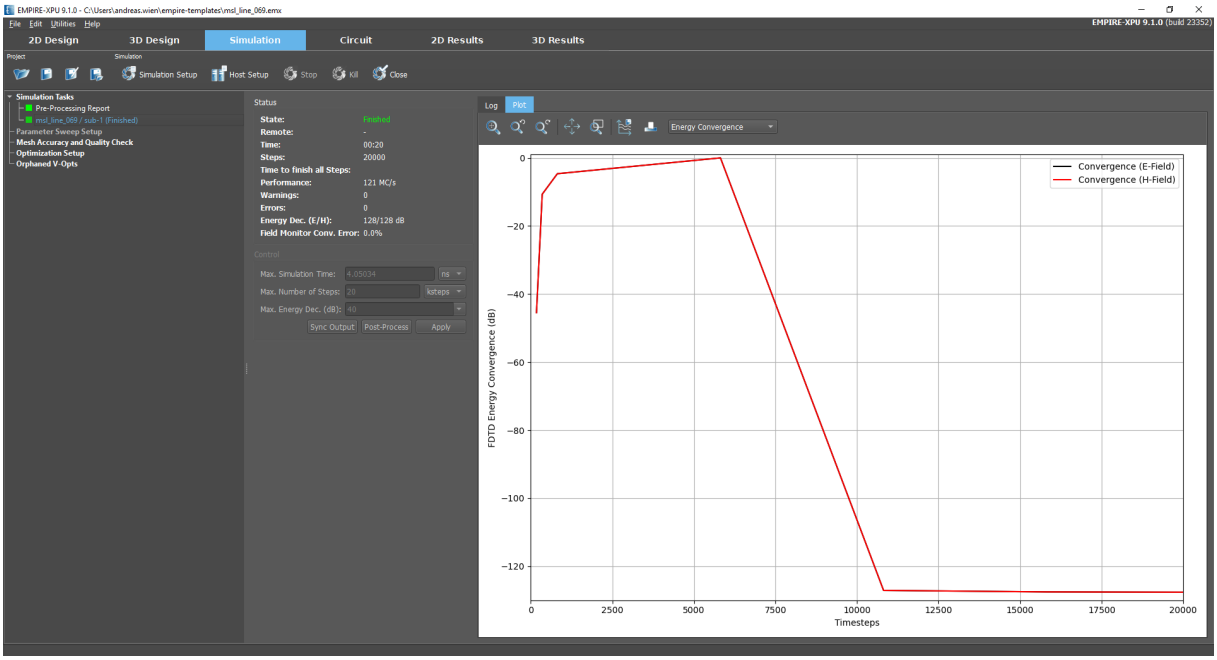


Figure 24.3.: Simulation Plot Window

24.4. Basic Simulation

The simulation can be started by clicking the **Start Simulation** button in one of the **Design** Tabs and and confirming the dialog. In the lower left area of the EMPIRE XPU™ window the progress of compilation is displayed. The simulation starts when the display mode is switched to **Simulation** tab where the evolution of the energy versus timesteps is depicted. The simulation is finished when the **State** is showing **Finished**.

In case of multiple sequential port excitations, the next port will be excited and a new simulation will be started automatically.

24.5. Advanced Simulation

If the simulation should be carried out in a batch mode or remotely, the user can switch to the **Simulation** Tab.

Clicking the button **Start Complete Simulation** has the same effect as in the **Basic Simulation**.

Clicking the button **Preprocessing** and confirming dialog boxes the window shows a list of project files and simulation folders which can be selected or de-selected.

The simulation is started for the selected folders by clicking the button **Start Simulation** . Then the status of the individual folders is displayed . During the simulation run, performance and status of the single simulation can be displayed by clicking on the job list on the left.

Available buttons for Simulation (Click on **Simulation Tasks** on left)

- **Simulation Setup** Parameters setup for simulation
- **Host Setup** Select hosts for the simulation
- **Start Complete Simulation** Carry out cleanup, preprocessing, simulation and postprocessing

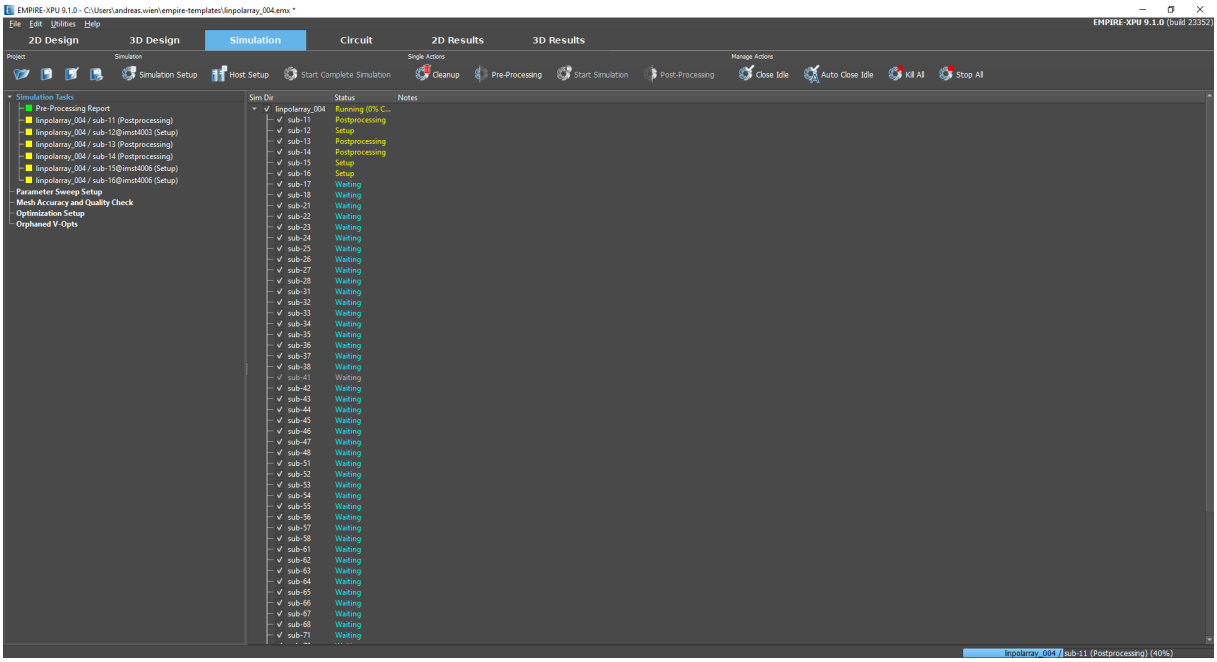


Figure 24.4.: Snapshot of Simulation Tasks

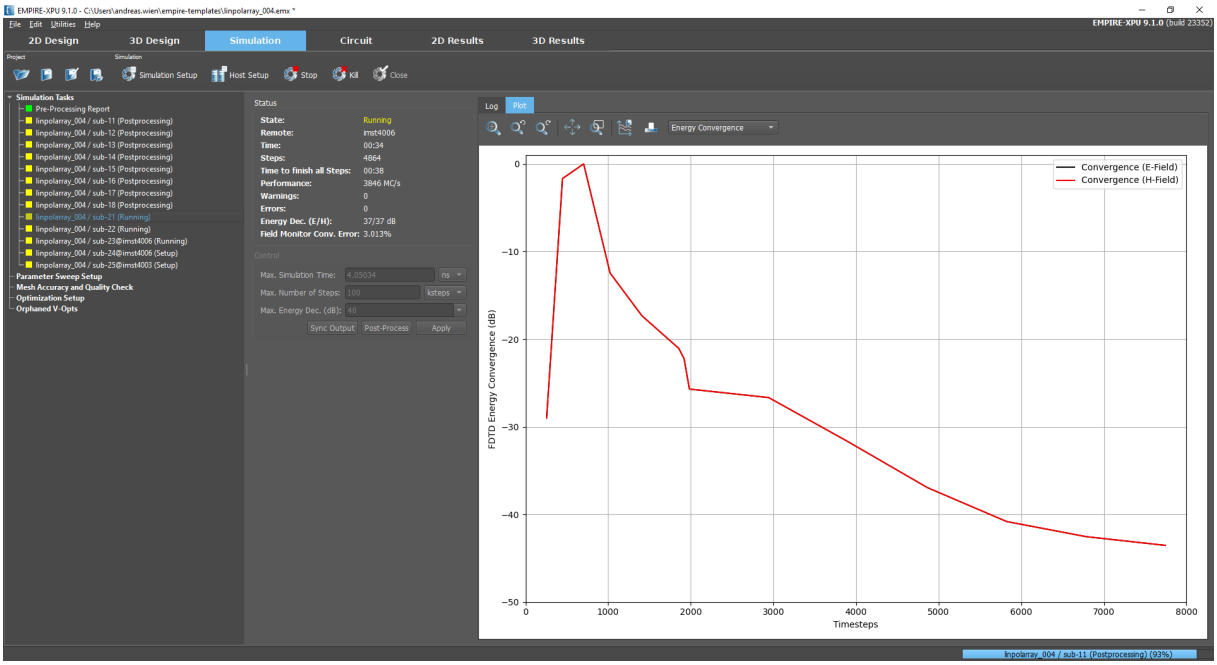


Figure 24.5.: Snapshot of a single job

- **Cleanup** Deletes all previous result files as defined in the **Cleanup setup**
- **Pre-Processing** Delete previous results and generate subfolders and data used for the simulation
- **Start Simulation** Start the simulation, including on-line code generation and FDTD algorithm. After the simulation time domain results are available
- **Post Processing** Start the postprocessing. (Re-)compute frequency domain data as well as far fields, equations, touchstone files or spice models if specified in the **Postprocessing** setup.
- **Close Idle** The status of the simulation(s) is displayed on the left side. Idle processes (green or red) can be closed to shorten the status list.
- **Auto Close Idle** The status of the simulation(s) is displayed on the left side. Idle processes (green or red) can be closed to shorten the status list.
- **Kill All** All running jobs will be aborted with no postprocessing.
- **Stop All** All running jobs will be aborted. Field dumps are generated and postprocessing will be executed.

During Simulation each job can be controlled by clicking on the respective sub folder entry on the left. It displays status and **Log** or **Plot** window of the running process.

Log and Plot Window (Select job on left)

Status

- **State:** Status of current simulation
- **Remote:** Name of remote host or - if local host
- **Time:** Current run time
- **Steps:** Already simulated time steps
- **Time to finish all Steps:** Time needed to reach the maximum number of time steps
- **Performance:** measured in million cells per second
- **Warnings:** Number of warnings (see Log Tab, yellow entries)
- **Errors:** Number of errors (see Log Tab, red entries)
- **Energy Dec. (E,H):** current energy estimate decay, for E and H fields
- **Field Monitor Convergence:** Accuracy of field monitors in frequency domain

Control

- **Max. Simulation time** End criteria in seconds
- **Max. Number of Steps** End criteria in timesteps
- **Max. Energy Dec. (dB)** End criteria defined by energy decay
- **Sync Output** Write out current near field data to get intermediate results for near and far fields
- **Post-Process** Start postprocessing of the running simulation
- **Apply** Set new values

It is possible to adjust the number of time steps during the simulation. This can be useful if the time signals have reached steady state before an end criterion has been reached, or if the signals don't decay fast enough so a larger number of time steps should be given. The expected end time (derived from the maximum number of time steps) is displayed in the monitor part of job control. For multi-port simulation, this time refers to the current port only.

Hint: Intermediate results

To obtain intermediate frequency domain results, a new job can be started by pressing the button **Post-Process**. DFT's of the current time signals will be performed and the scattering parameters will be calculated without stopping the running simulation. If intermediate near and far fields should be obtained, press the button **Sync Output** first.

24.6. Batch processing

If several jobs should be queued they can be added with right click on **Simulation Tasks**, selecting **Add external Simulation** and search for an available pre-processed file **.acad**.

For all jobs which should be simulated the preprocessed files **.acad** have to exist in the respective folders.

The folders will be added to the list and can be selected in order to simulate

- one after another on one machine
- in parallel on a PC cluster

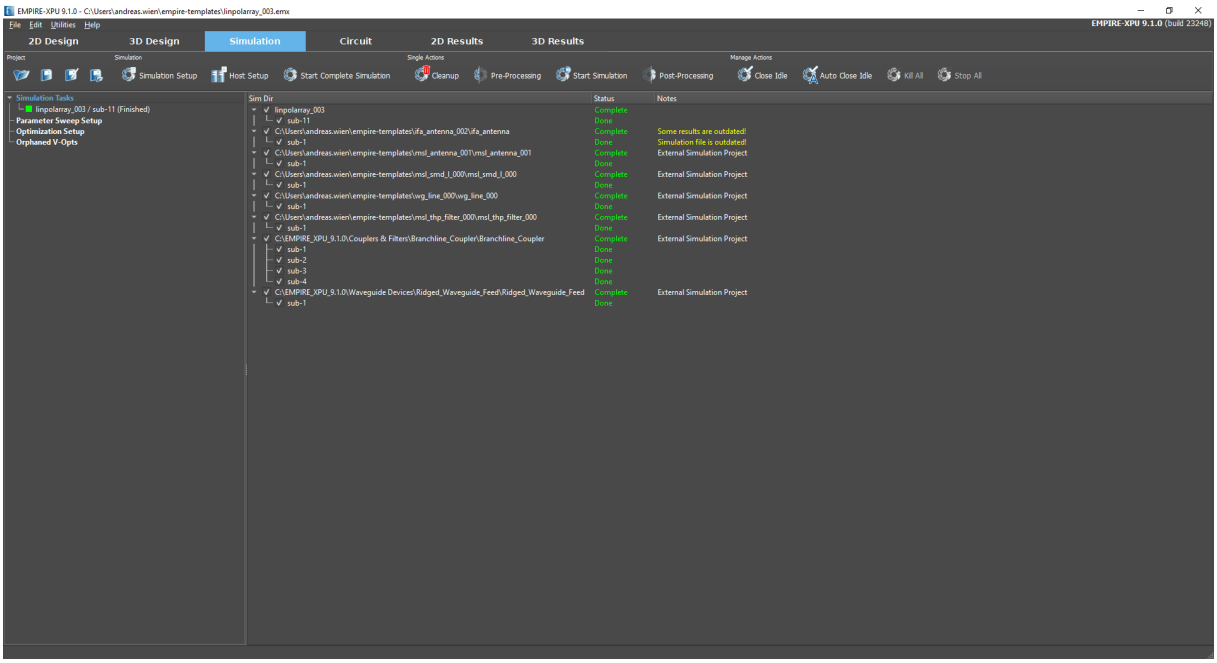


Figure 24.6.: Batch control window

In order to envoke other PCs see chapter 24.10

Hint: Application Note

More information is available in Chapter 29.2.1

24.7. Parameter Sweep

If variables have been defined, they can be be varied automatically depending on their range and step values defined in the **Variables** entry in one of the **Design** tabs. Simulations will be started for the selected variable combinations either subsequently on a single PC or parallel on multiple PC which are selected in the **Host Setup**. Each variation creates a unique folder starting with **v-opt** followed by a number, parameter names and values.

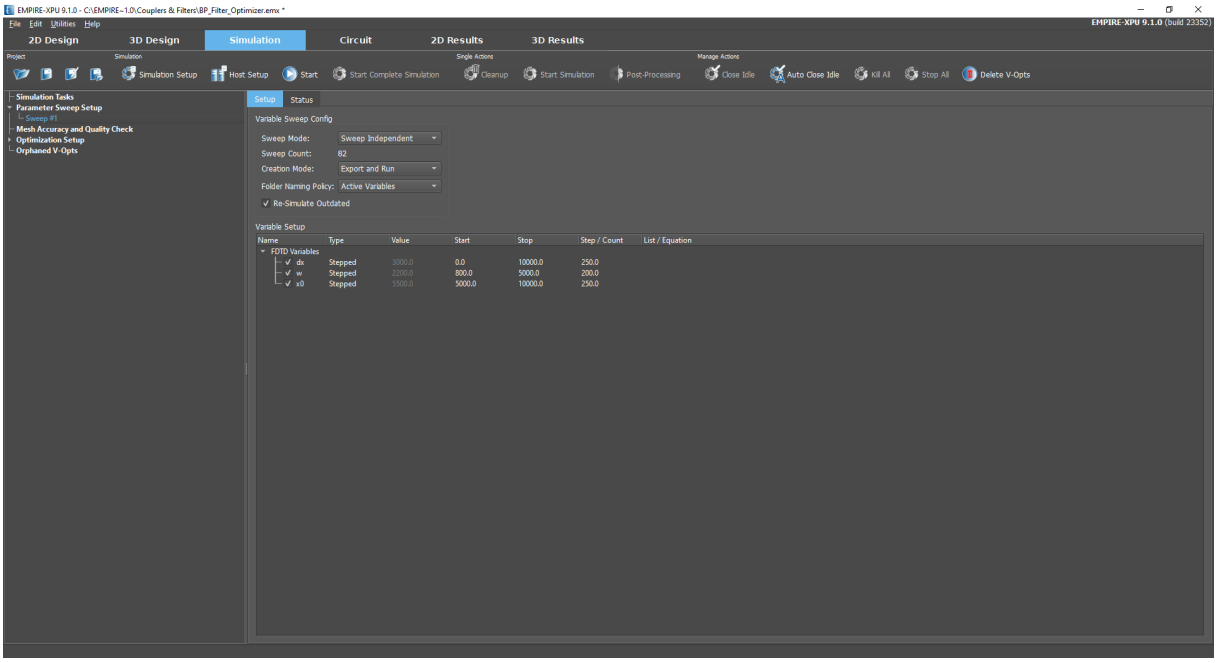


Figure 24.7.: Parameter sweep control window

Control

- Click on **Parameter Sweep Setup** on the left
- Click button **Create Parameter Sweep** to add one or more parameter sweep controls
- Click on **Sweep(n)** entry on the left

A frame with **Setup** and **Status** tabs are now visible.

Parameter Sweep Buttons

- **Start** - Start parameter sweep process
- **Cleanup** Deletes all previous result files
- **Postprocessing** - Start postprocessing only (during or after the simulation)
- **Close Idle** - Close status display on the left of finished simulations

- **Auto Close Idle** The status of the simulation(s) is displayed on the left side. Idle processes (green or red) can be closed to shorten the status list.
- **Kill All** All running jobs will be aborted with no postprocessing.
- **Stop All** Running jobs will be aborted. Field dumps are generated and postprocessing will be executed.
- **Delete V-Opts** - Delete previous results

Setup - Variable Sweep Config

If many parameters and steps have been defined the number of combinations (and so the number of simulations) can be very large.

- **Sweep Mode** - Parameter sweep method (n is the number of parameters and k_i is the number of steps of each parameter)
  - **Sweep Independent** This mode keeps all but one variable fixed and sweeps the parameters independently which leads to $1 + \sum_{i=1}^{n}(k_i - 1)$ simulations. Used by default.
  - **Sweep parameter space** This mode simulates all possible parameter combinations $\prod_{i=1}^{n} k_i$ which quickly becomes huge.
- **Sweep Count** displays the total number of simulations
- **Creation Mode**
  - **Create Sweep only** Creates sweep plan in status window
  - **Export only** Additionally, creates preprocessed files in variation folders
  - **Export and Run** Additionally, simulates in each created folder
- **Folder Naming Policy** - In case of many parameters or long parameter names, the folder names can become longer than permitted by the operating system. The value can be set to shorten the names if necessary

Variable Setup

All variables are listed with their value, range and step or list. Their values can be set by double-click. They may be deactivated by clicking on the respective checkmark. The sweep count is updated accordingly.

Status Tab Window displaying current variable combinations, simulations and progress of the sweep.

Hint: Application Note

More information is available in Chapter 29.2.17

24.8. Mesh Accuracy and Quality Check

This special sweep mode enables to perform mesh convergence studies which allow to check the accuracy and quality of the adaptive hexahedral mesh using different refinement strategies.

- **Reference Simulation**
  No refinement; initial mesh as defined in **Simualtion Setup - Mesh**
- **Hard XYZ-Mesh Refinement**
  Subdivide each cell of the initial mesh by a factor (1= no refinement, 2= divide cells in half, etc.). This refinement can be limited to different directions x,y,z and to regions, see Section 18.5.2.
- **Automesh Expert-Mesh Refinement**
  Stepwise increase the **Mesh Resolution** which is defined in **Simulation Setup - Mesh**, see Section 22.1.3.

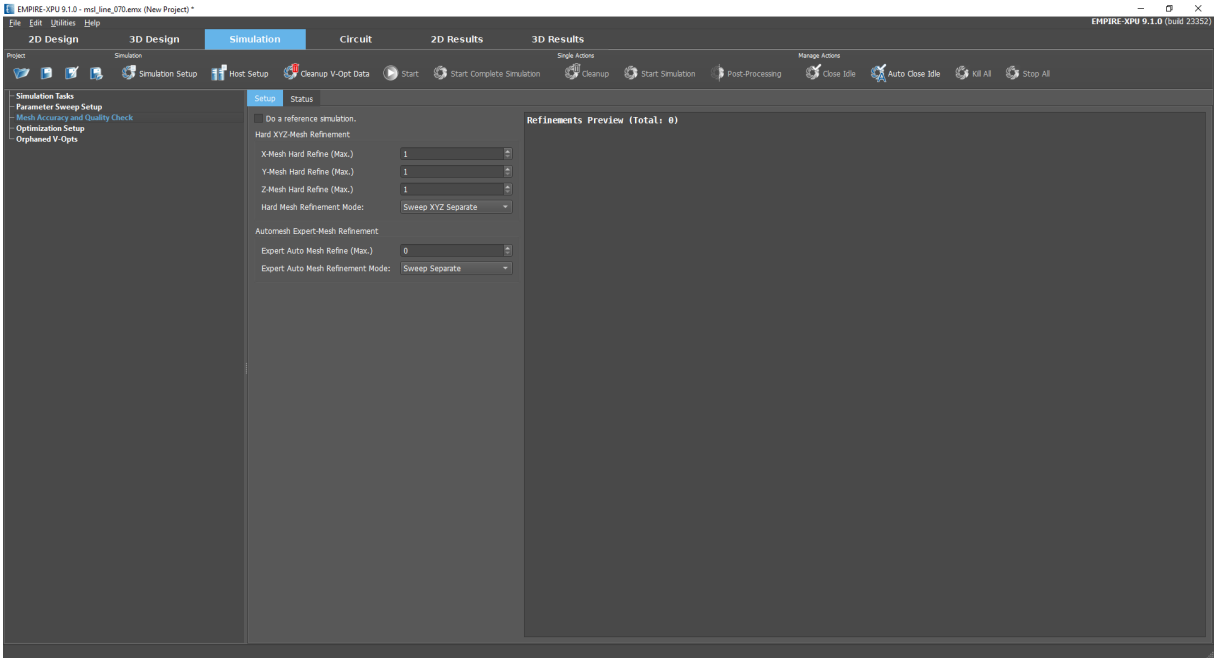


Figure 24.8.: Optimizer control window

24.8.1. Hard XYZ-Mesh Refinement

- Hard mesh refinement by a fixed factor can be defined and performed separately in the three different directions (x,y,z)
- Hard refinement is a requirement, e.g., for compliance simulations in accordance with IEC 62704-1
- The refinement takes place in successive steps; this is controlled by the user under **Hard Refinement Mode**
  - **Sweep XYZ Separate** Keep 2 directions fixed at lowest value and sweep 1 direction only.
  - **Sweep Combined Max. Only** Keep 2 directions fixed at highest value and sweep 1 direction only.
  - **Sweep Completely** All possible combinations
- The hard mesh refinement can be limited to regions, defined by objects on a group with property **Mesh Accuracy and Quality Check** see Section 18.5.2.
- Under 2D & 3D Design the user will find three new stepped variables, defined with their correct ranges, namely **mesh_refine_x (y, z)**
- While not simulating, the user can adjust the sliders at will and re-mesh the model, so as to get an idea of the achieved mesh refinement
- After sweep, the convergence can be checked for each result in the **2D Results** tab as a function of variables **mesh_refine_x (y, z)**

24.8.2. Automesh Expert-Mesh Refinement

- By increasing the integer numerical input to N, a total of N-1 trials are listed under **Refinements Preview**
- Each trial is assigned an integer descriptor, e.g., 1, 5, 10,..., 30 which correspond to a unique mesh resolution as listed in Table 22.1.3
- At the same time, a variable is automatically generated under **2D Design** : mesh_refine_auto
- To preview the mesh refinement, the variable slider can be shifted to a new resolution and by clicking on **Create Mesh** the mesh is displayed
- **mesh_refine_auto** is a global mesh variable, that is, it acts on the spatial grid of all three Cartesian axes at once
- This refinement can be swept separately or in conjunction with **Hard Mesh Refinement**
- After sweep, the convergence can be checked for each result in the **2D Results** tab as a function of variable **mesh_refine_auto**

24.9. Optimization

If objects or properties have been defined with variables, they can be optimized according to a specific goal. The optimizer applies a certain strategy to find the optimum variable set following the entered optimization goal and terminates the process if the goal has been reached or the optimum has been found.

Each optimization step creates a unique folder starting with **v-opt** followed by a number, variable names and values.

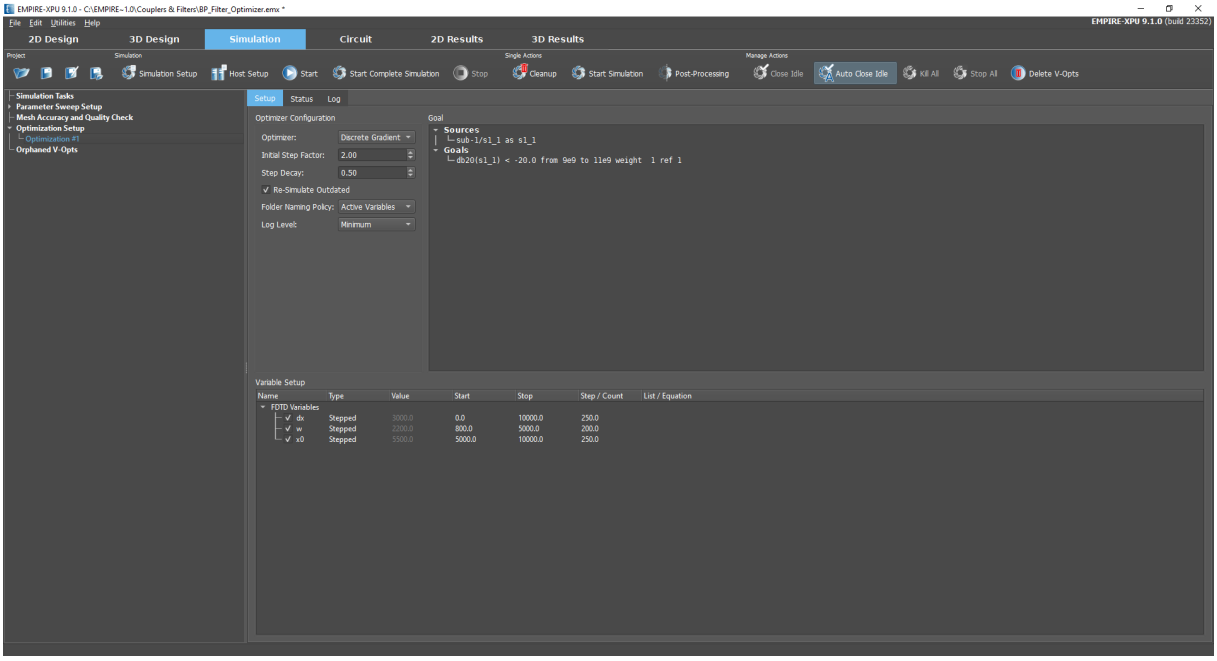


Figure 24.9.: Optimizer control window

Hint: Application Note

More information is available in Chapter 29.2.16

24.9.1. Optimization Control

- Click on **Optimization Setup** on the left
- Click button **Create Optimization** to add one or more optimization controls
- Click on **Optimization(n)** entry on the left

A frame with **Setup**, **Status** and **Log** tabs are now visible.

Optimization Buttons

- **Start** - Start parameter sweep process
- **Stop** - Stop parameter sweep process
- **Postprocessing** - Start postprocessing only (during or after the simulation)
- **Close Idle** - Close status display on the left of finished simulations
- **Auto Close Idle** The status of the simulation(s) is displayed on the left side. Idle processes (green or red) can be closed to shorten the status list.
- **Kill All** All running jobs will be aborted with no postprocessing.
- **Stop All** Running jobs will be aborted. Field dumps are generated and postprocessing will be executed.
- **Delete V-Opts** - Delete previous results

24.9.2. Optimization Configuration

- **Optimizer** Chose optimization algorithm
  - **Discrete Gradient** - Gradient optimization algorithm. Searches by locally iterating through the points next to the start point. The best becomes the new start point, until a local optimum is found.
  - **Nelder-Mead** - Efficient local optimization using a simplex algorithm. This popular, simple, robust and deterministic local optimizer performs well – occasionaly even for noisy behavior. There are theoretical situations where is doesn't work though.
  - **BOBYQA** - nonlinear derivative-free bound constrained optimization method. BOBYQA is a local optimizer based on quadratic approximation. It speeds to local optima near the startpoint. The basic assumption is that the behavior is twice differentiable, so problems with strong nonlinearity can cause failure.
  - **GN Direct** - Global optimization, parallelized. The Dividing RECTangles algorithm is a deterministic broad search. Suitable for problems where top-down search leads to a fast solution, e.g. if there are global trends in the parameters.
  - **MLSL** - Multi-level single-linkage random optimizer. Searches around random points with local D.G. optimization. A heuristic influences the start points, so that all local optima can be found. The performance balances the broad search and local search advantages.

  - **L-BFGS-B** Limited-memory BFGS (L-BFGS or LM-BFGS) is an optimization algorithm in the family of quasi-Newton methods that approximates the Broyden–Fletcher–Goldfarb–Shanno (BFGS) algorithm using a limited amount of computer memory. It is a popular algorithm for parameter estimation in machine learning.
  - **TNC** Truncated Newton method, also known as Hessian-free optimization is an optimization algorithm designed for optimizing non-linear functions with large numbers of independent variables. A truncated Newton method consists of repeated application of an iterative optimization algorithm to approximately solve Newton's equations, to determine an update to the function's parameters.
  - **SLSQP** SLSQP optimizer is a sequential least squares programming algorithm which uses the Han–Powell quasi–Newton method with a BFGS update of the B–matrix and an L1–test function in the step–length algorithm. The optimizer uses a slightly modified version of Lawson and Hanson's NNLS nonlinear least-squares solver
  - **Random Uniform** Simple random method. The Optimizer tests random parameter combinations until it satisfys the condition or tested every point. The performance is poor but convergence to the global optimum is assured.

- **Initial Step Factor** - Step enlargement for first iteration
- **Step Decay** - Step decay for next iteration
- **Sequential Subdir Calculation** - Used for result dependent optimization
- **Re-Simulate Outdated** Choose if old results should be used or not.
- **Folder Naming Policy** - In case of many parameters or long parameter names, the folder names can become longer than permitted by the operating system. The value can be set shorten the names.
- **Log Level** - Specifies details of log output during optimization

The optimization process can be described as follows:

- The optimization starts with the initial **step factor** times the **step size** defined in parameters
- During the optimization the deviation from the goal function will be determined
- If the goal could not be achieved the **step factor** will be multiplied by the **decay factor** and a new iteration will be started
- If the **step size** is reached, the optimization process will be terminated and the best solution achieved so far will be regarded as optimum.

Variable Setup

All variables defined in the **Design** Tabs are listed with their value, range and step or list.They may be deactivated for the current optimization by clicking on the respective checkmark.

24.9.3. Optimization Goal

One or multiple weighted goals can be defined for optimizations. They can be added by right click in the **Goal** frame and edited, copied or removed by right click on the respective item. Supported result data files are:

- **2D Results** Multiple value data files with frequency, angle, or distance dependence (e.g. s1_1, Farfield_..._ephi, f_dump_..._ex, ...)
- **1D Results** Single value log files (e.g. Farfield_..._ephi.log) with specific results (e.g. efficiency RADEFF, maximum MAX, ...)[2]

[2] If far fields have been calculated a set of results will be written into log files. These "single value" results can be used for optimization with **Add Log Goal**

In order to compose one or more goals file names and locations have to be specified. For **2D Results** both **Source** and **Goal** have to be added while for **1D Results** only a **Log Goal** deinition is sufficient.

Sources (for 2D Results, only)

- Right click in Goal frame and select **Add Source**
- **Data Folder** Location of the result file
- **File Name** Result files which are dependent on frequency or angle to be used for goal definition.
- **As** Short name for the source to be used in goal equation
- **Browse** Search for a source file of an already available simulation folder. (It is good practice to run a single simulation before setting up the goal and run the optimization. Then the far field source files can be selected easily and do not have to be entered manually.)

Goal (for 2D Results, only)

- **X Range** - Start and end value of target interval, e.g. $f_0 - f_1$ or $\theta_0 - \theta_1$
- **Normalization** - Reference value used for error function to normalize expressions with different units.
- **Weight** - Value used to weight different goals.
- **Equation** - Goal function using the short name for data files. A list of supported functions (e.g. imag(), angle()) is available at `https://numpy.org/doc/stable/reference/routines.math.html`. A special function dB20() is available for S-parameters.

Log File Goal (for 1D Results, only)

- **Log file** - A list of available log files is contained in the drop down list
- **Normalization** - Reference value used for error function to normalize expressions with different units.
- **Weight** - Value used to weight different goals.
- **Equation** - Goal function for parameters available in the drop down list.

Hint: Deactivate Goal

In order to temporarily deactivate a goal just apply a weight factor of 0. Make sure that at least one goal weight is different to zero.

During optimization the deviation from the goal function is calculated as follows:

$$E = \sqrt{\frac{\sum_{i=1}^{m} \frac{weight_i}{n} \sum_{k=1}^{n} (\frac{expression_{i,k} - goal_i}{ref_i})^2}{\sum_{i=1}^{m} weight_i}}$$

Here m is the number of goals and n is the number of parameter points.

Status Tab Window displaying current variable combinations, simulations and progress of the optimization.

Log Tab Window displaying current error and progress of optimization.

A log file `optim_<date>_<time>.log` is created in the project folder which contains optimizer statistics.

24.9.4. Optimization Goal Examples

Reflection matching

▼ **Sources**

```
├ sub-1/s1_1 as s11
```

▼ **Goals**

```
├ db20(s11) < -15.0 from 1.0e+010 to 1.1e+010 weight 1 ref 1
```

Phase difference (radiant)

▼ **Sources**

```
├ sub-1/s2_1 as s21
├ sub-1/s3_1 as s31
```

▼ **Goals**

```
├ abs(angle(s21)-angle(s31)) < 0.1 from 5e+9 to 6e+9 weight 1.0 ref 1.0
```

Farfield Pattern

▼ **Sources**

```
├ sub-1\Directivity_Farfield_1_f1.00000e+10_p0.000e+00_eabs as ff
```

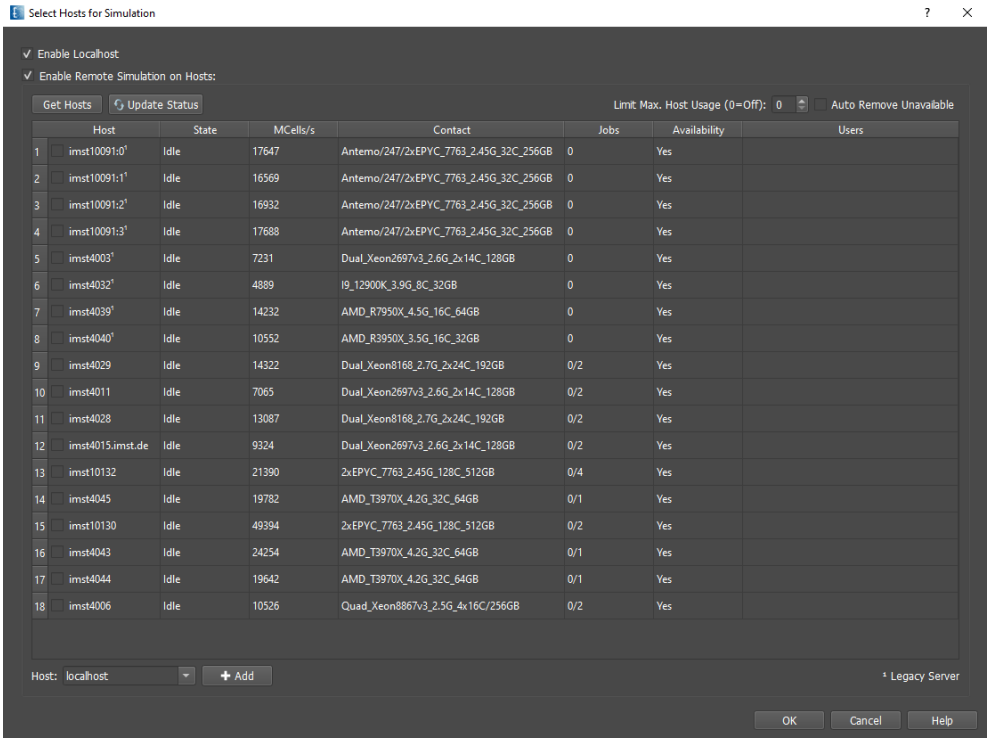
▼ **Goals**

```
├ db20(ff) < 0.0 from -60 to -30 weight 0.5 ref 1.0
├ db20(ff) > 10.0 from -5 to 5 weight 1.0 ref 1.0
├ db20(ff) < 0.0 from 30 to 60 weight 0.5 ref 1.0
```

Farfield 3dB Bandwidth

▼ **Goals**

```
├ sub-1\Directivity_Farfield_1_f1.00000e+10_p0.000e+00_eabs.log W3DB < 30.0 weight 1.0 ref 1.0
```



	Host	State	MCells/s	Contact	Jobs	Availability	Users
1	imst10091:0¹	Idle	17647	Antemo/247/2xEPYC_7763_2.45G_32C_256GB	0	Yes	
2	imst10091:1¹	Idle	16569	Antemo/247/2xEPYC_7763_2.45G_32C_256GB	0	Yes	
3	imst10091:2¹	Idle	16932	Antemo/247/2xEPYC_7763_2.45G_32C_256GB	0	Yes	
4	imst10091:3¹	Idle	17688	Antemo/247/2xEPYC_7763_2.45G_32C_256GB	0	Yes	
5	imst4003¹	Idle	7231	Dual_Xeon2697v3_2.6G_2x14C_128GB	0	Yes	
6	imst4032¹	Idle	4889	I9_12900K_3.9G_8C_32GB	0	Yes	
7	imst4039¹	Idle	14232	AMD_R7950X_4.5G_16C_64GB	0	Yes	
8	imst4040¹	Idle	10552	AMD_R3950X_3.5G_16C_32GB	0	Yes	
9	imst4029	Idle	14322	Dual_Xeon8168_2.7G_2x24C_192GB	0/2	Yes	
10	imst4011	Idle	7065	Dual_Xeon2697v3_2.6G_2x14C_128GB	0/2	Yes	
11	imst4028	Idle	13087	Dual_Xeon8168_2.7G_2x24C_192GB	0/2	Yes	
12	imst4015.imst.de	Idle	9324	Dual_Xeon2697v3_2.6G_2x14C_128GB	0/2	Yes	
13	imst10132	Idle	21390	2xEPYC_7763_2.45G_128C_512GB	0/4	Yes	
14	imst4045	Idle	19782	AMD_T3970X_4.2G_32C_64GB	0/1	Yes	
15	imst10130	Idle	49394	2xEPYC_7763_2.45G_128C_512GB	0/2	Yes	
16	imst4043	Idle	24254	AMD_T3970X_4.2G_32C_64GB	0/1	Yes	
17	imst4044	Idle	19642	AMD_T3970X_4.2G_32C_64GB	0/1	Yes	
18	imst4006	Idle	10526	Quad_Xeon8867v3_2.5G_4x16C/256GB	0/2	Yes	

Figure 24.10.: Hosts control window

24.10. Remote Processing

By default, the local host will be used for simulation. If EMPIRE XPU™ is installed on other machines in the network and the EMPIRE XPU™ **Utilities - Server** (reconnectable) or **Utilities - Server (Legacy)** has been started on them, they can be used to run the simulation remotely. This works for Linux and Windows and cross installations.

- **Enable Localhost** - Use local machine for simulation
- **Enable Remote Simulation on Hosts:** - Use local machine for simulation
- **Get hosts** - Get a list of all available hosts in the LAN
- **Update Status** - Refresh list of available hosts
- **Limit Max. Host Usage** - Host number limitation if multiple hosts selected
- **Auto Remove Unavailable** - List only currently available hosts
- **Columns**
  - **Host** - Hostname in network (superscript 1 denote **Legacy Server**)
  - **State** - **busy** if host is processing a job
  - **Mcells/s** Performance of host
  - **Contact** - Information of owner, RAM size, etc.
  - **Jobs** - Number of currently running jobs on this host

  - **Availability** - Yes / No
  - **Users** - Owners of currently running jobs

- **Host** - Name of the processor to be added to hostlist
- **Process Number limit** - Specify number of jobs which can be handled by this processor in parallel.
- **Add** - Select specific host or processor to the host list

If the EMPIRE XPU™ remote service is started on the local machine it will be available in the host list, too. To avoid parallel simulation on the local machine, the **Localhost** should be unchecked.

Hint: Application Note

More information is available in Chapter 29.2.19

24.11. Post Processing

After the simulation has finished the last time step, the postprocessing is started automatically which includes:

- **Wave separation** - Incident and reflected time signal of excitation
- **DFT** - Discrete Fourier Transformation of time signals
- **Frequency domain results** - Impedance, S-Parameter, Power calculation
- **Far field calculation** - Near field to Far field transformation

It is possible to start subsequent postprocessings by clicking the **Post-Processing** button . This is useful if only postprocessing parameters are changed and a full new simulation is not required (e.g. change of frequency range, signal estimation order, far field angle resolution, far field normalization, ...)

25. Results

After simulation and postprocessing all results like S-parameters or field dumps are available in the result folders sub-i (where i is the exciting port number) to be displayed or further processed.

- **Single Values** of far field transformations like radiated power, efficiencies, 3 dB beamwidth, etc, are printed in associated .log files and can be further processed with the optimizer. These values can also be plotted in the **2D Results** tab using the **Plot Type: Far Fields Meta Data**
- **2D plots** can be displayed in the **2D Results** Tab, e.g. **Voltage** vs. time (ut1), **S-Parameter** vs. Frequency (s1_1) **Impedance** vs. frequency (Z1.in), **Farfield** vs. single angle (Farfield_frequency_angle_component). By selecting the **Plot Type** the different results can be displayed and multiple frames can be used.
- **3D near field** (emvolume.dbx) can be displayed in the **3D Results** Tab together with structure and controlled with the right click on field monitor entries.
- **3D far field** (Farfield_frequency_component) can be displayed in the **3D Results** Tab together with structure and controlled with a right click on the field monitor entries. In **2D Results** Tab the 3D far field can be displayed in a as a color map by selecting the plot type **Far Fields (3D)**.
- **3D Temperature Distribution** (th.dat) can be displayed in the **3D Results** Tab and controlled with a right click on the thermal field monitor

Hint: Results

All frequency domain results are RMS (root mean square) values.

Hint: Spherical coordinates

In case of 2D plots for the far field results, the angle φ refers to the angle between x axis and y axis and θ refers to the angle between z axis and xy-plane.

25.1. 2D Results set up

By default one graph frame is opened in the **2D Results** tab. More frames can be opened with **Add Result** button.

The **Plot Type** and **Plot Format** can be changed for other result types or formats and saved for the current project. The **2D Results** areas are:

- **Button Bar**: Collection of actions for the active graph frame
- **Graph Frame**: Displaying curves with labels; right click for context menu
- **Plot Type and Format**: Setting type and format of the result files to be displayed
- **Results**: List and control of curves to be displayed; right click for context menu
- **Marker**: List of markers if set; click on marker buttons to add marker, e.g.
- **Masks**: List of masks if set; right click to add mask

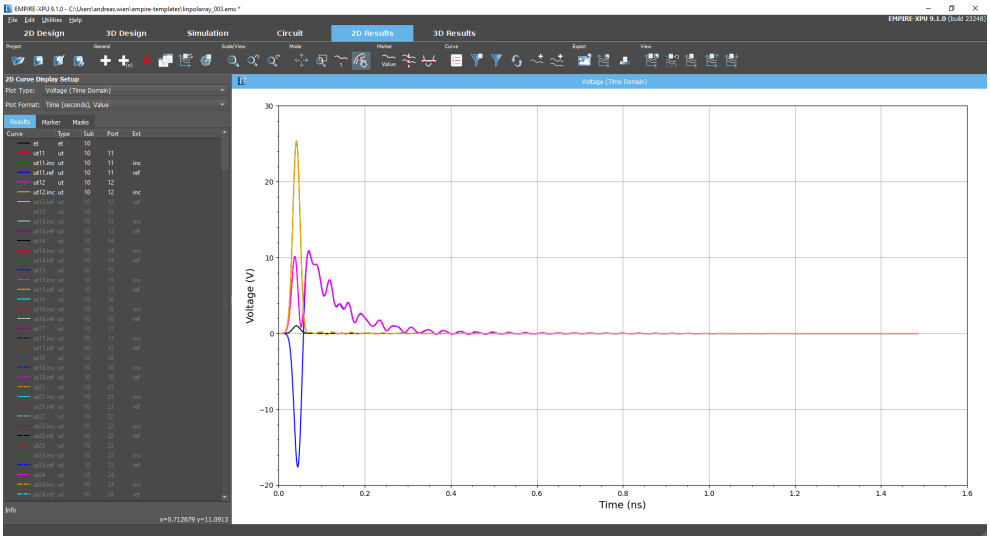


Figure 25.1.: 2D graph display for voltage

25.1.1. Button Bar

In the top area there are buttons for the following actions.

- **Project**
  - **Open Project** Load another project
  - **Save** Save current project
  - **Save As** Save current project to new folder
  - **Store Variant** Save current project with results to new folder
- **General**
  - **Add Result** Open a graph frame for **2D Curve Display Setup**
  - **Add Sweep Result** Open a graph frame for **Variable Sweep Curve Setup**
  - **Remove Result** Close active graph frame
  - **Toggle View Mode** Arrange frame in tabs or tiles
  - **Tile Sub Windows** If tile is active, rearrange frames
  - **Reset to Default View Settings** Undo view settings
  - **Global Plot Configuration** Adjust fontsize, grid or label appearance
- **Scale/View**
  - **Autoscale** shows the complete range for both axes
  - **Previous View** steps back to last views
  - **Next View** steps forward to last views
- **Mode** Selected mode is active until another mode is clicked
  - **Pan/Zoom** Change view

    - Drag left mouse in plot area: dynamic shift of the plot
    - Drag right mouse button: dynamic zoom of the plot
  - **Zoom Rect** creates a zoomed view defined by a rectangle in rectangular plots
  - **Place Marker** after pressing, a marker can be positioned with cursor
  - **Pick Curve/Marker/Legend** Select line, marker or legend with the cursor. The respective item is highlighted in the list
- **Marker**
  - **Toggle Marker Text** Display or hide marker information
  - **Place Multiple Marker (X)** Places markers on all curves at a given X value
  - **Place Multiple Marker (Y)** Places markers on all curves at a given Y value
- **Curve**
  - **Toggle Legend** Displays the legend inside of the graph. Its position can be shifted with the mouse cursor
  - **Filter Settings** Define a filter for the reduction of the result list
  - **Enable Filter** Apply or unset filter
  - **Reload Data** Update result database
  - **Add Data File** Open the windows explorer to add a file to the legend area
  - **Add Data Directory** Open the Windows explorer to select a folder and add matching results to the legend area
- **Export**
  - **Export Image** generates an image for plot and legend
  - **Export Plot Data** generates a .csv file of the diaplyed plot
  - **Print Plot** Creates a picture of the graph and invokes an associated program for printing
- **View Settings**
  - **Quick Save View Settings** Save the current settings
  - **Quick Load View Settings** Restore settings which have been saved
  - **Delete Quick View Settings** Remove settings which have been saved
  - **Export Settings** Save the current settings in a specified file
  - **Import Settings** Restore settings from a specified file

25.1.2. Graph Frame

In this area the results depending on the adjustments in **Plot Type** are displayed. By default all results in the current project folder which fit to the selection are detected and listed in the **Results** tab on the left. Becasue the list may be very long, only the first 5 files will be displayed in the plotting area while the rest are hidden by default. With a right click selected or all curves can be added.

Hint: Copy to Clipboard

With **Ctrl-c** the graph is copied to the clipboard and can be pasted into documents (e.g. Word) with **Ctrl-v**

Plot Configuration

Labels and Ranges can be changed by a right click in the graph frame and selecting **Configure Plot**. The following menu appears

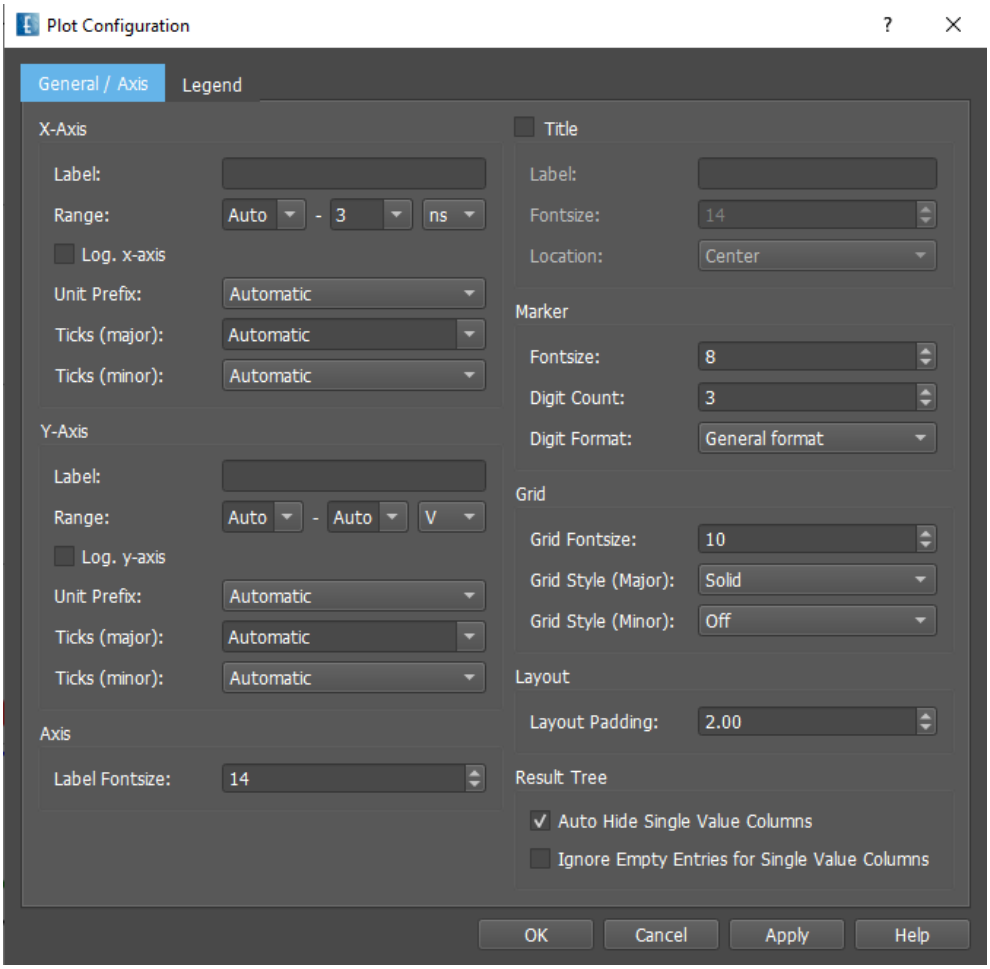


Figure 25.2.: Global plot settings (right click in plot area)

General / Axis Tab

For the respective axis the following can be adjusted on the left

- **Label** Set label (x Bottom, y Left)
- **Range** Plot range
- **Log. x/y-axis** Plot axis in logarithmic format
- **Unit Prefix** Label unit
- **Ticks (major)** Number of Plot grid lines
- **Ticks (minor)** Number of Plot sub grid ticks
- **Label Fontsize** Applies to both labels

On the left, format and style can be set

- **Title** Heading at top of the graph
- **Marker** Size, number of digits and format
- **Grid** Label and line style
- **Layout** Space to borders
- **Result Tree** In order to keep the result tree clearly arranged options for single valued columns can be set.

Legend Tab

- **Enable** The legend can be enabled with a checkmark, same as 
- **Fontsize** Size of legend characters
- **Location** Position of legend in graph
- **Show Full Path** Location of result file relative to project folder
- **Show Parameter** Include variable names
- **Use Format Prefix** Include prefix if applicable
- **Show Mask Labels** Include mask if applicable

The entries in **Legend Label** can also be edited with a double click in the list.

25.1.3. 2D Curve Display Setup - Plot Type

In this frame the different plot results can be selected. Depending on the selected **Type** the project result folders will be browsed for all matching files and displayed in the legend and graph.

- **Voltage (Time Domain)** applies to ut* and et files
- **Voltage (Frequency Domain)** applies to uf* and ef files
- **Current (Time Domain)** applies to it* files
- **Current (Frequency Domain)** applies to if* files
- **Scattering Parameters** applies to s* files
- **Impedance** applies to Z* files
- **Impedance (TDR)** applies to Z* files for Time Domain Reflectrometry applications
- **Admittance** applies to Y* files
- **Power (Frequency Domain)** applies to P* files
- **All YZ (Frequency Domain)** Y*, Z* files

- **Far Fields** 2D Far field results with cut angle
- **Far Fields Meta Data** 1D Far field results as available in associated log files
- **Far Fields (3D)** 3D Far field results
- **Far Fields (RCS)** Bi-static scattering cross sections
- **Antenna Efficiency** versus frequency
- **Scattering Cross Section** RCS over frequency

- **Far Fields (Frequency Domain)** Far field results of fixed angles over frequency
- **MIMO ECC** Envelope Correlation Coefficient **ECC** (if enabled in Far field Monitor - Mimo Correlation Setup)
- **MIMO EDCF** Envelope Diagram Correlation Factor **DCF** based on S-Parameters (if enabled in Far field Monitor - Mimo Correlation Setup)

- **Field Monitor Convergence** Error of frequency domain field monitors versus time steps
- **Energy Convergence** Energy in simulation domain versus time steps
- **Speed Log** Performance versus time steps

- **E-Probe (Frequency Domain)** Electric field over frequency
- **E-Probe (Time Domain)** Electric field over time
- **H-Probe (Frequency Domain)** Magnetic field over frequency
- **H-Probe (Time Domain)** Magnetic flux over time
- μ_0***H-Probe (Frequency Domain)** Magnetic flux over frequency
- μ_0***H-Probe (Time Domain)** Magnetic field over time
- **E-Field Path** Electric field along a path
- **H-Field Path** Magnetic field along a path
- **S-Field Path** Poynting vector along a path
- **Surface Power Density (Meta Data)** 1D PD results as available in associated log files
- μ_0***H-Field Path** Magnetic flux along a path

- **EIAV Distribution** Histogram of averaged internal electric field inside body models

- **Y Parameters** Admittance parameters in ypar folder
- **Z Parameters** Impedance parameters in ypar folder
- **ABCD parameters** applies on ab* files
- **Image parameters** applies on I* files
- **Noise Parameters** applies on F* files
- **Stability Factor, k(f)** applies on k* files
- **Stability Factor, mu(f)** applies on mu* files

- **Custom Result** user defined results with arbitrary units. Files can be added with **Add Data File** button.
- **Legacy Custom Curve, (FD)** user defined frequency domain results, e.g. postprocessing results created with a **Symbolic Equation**. Files with no label can be added with **Add Data File** button. If a label (and associated **.label** file) exists the curves will be listed and can be filtered as usual data.

25.1.4. 2D Curve Display Setup - Plot Format

Depending on the selected **Type** only some of the following formats are available.

Time Domain

- **Time (seconds) Value** = Display of real type values in seconds
- **Time (seconds) Lin. Magnitude** = Display of absolute values in seconds
- **Time (seconds) Log. Magnitude** = Display of logarithmic absolute type values in seconds
- **Time (steps)** Using timestep instead of seconds
- **Electric length (m)** Using electric length instead of seconds

Frequency and Angular Domain

- **Lin. Magnitude** = $|z(x)|$ Magnitude in linear style
- **Log. Magnitude** = $20 \log |z(x)|$ Magnitude in logarithmic style
- **Abs. Log. Magnitude** = $|20 \log |z(x)||$ Magnitude in logarithmic style

- **Angle (deg)** = $\varphi(f) = \frac{180}{\pi} \arctan \frac{\Im z}{\Re z}$
- **Group Delay** = $\frac{1}{360} \frac{d\varphi}{df}$
- **Group Delay (el.length)** = $\frac{c_0}{360} \frac{d\varphi}{df}$
- **Angle Unwrapped (deg)** = $\varphi(f)$ avoiding 360 degree jumps

- **Lin. Magnitude / Angle (deg)** - Linear magnitude and angle in degrees of a complex value
- **Log. Magnitude (dB) / Group Delay (e. length)** - Logarithmic magnitude and group delay of a complex value
- **Log. Magnitude (dB) / Angle (deg)** - Logarithmic magnitude and angle in degrees of a complex value

- **Re** - Real part of a complex value
- **Im** - Imaginary part of a complex value
- **Re/Im** - Real and imaginary part of a complex value
- **Mag/Re/Im** - Real and imaginary part and magnitude of a complex value

Additional for S-Parameters

- **VSWR** Voltage Standing Wave Ratio
- **Smith Chart** for S-parameters with reference impedance of 50Ω
- **Polar** Polar diagram for S-parameters

Additional for Far fields

- **Polar Lin. Magnitude** Linear far field diagram
- **Polar Log. Magnitude (dB)** Logarithmic far field diagram

25.1.5. 2D Curve Display Setup - Results

In the **Results** Tab the files are listed and columns are available which can be used for sorting and filtering of the files. The number of columns depends on the **Type** selection and sweep parameters if in use.

Hint: Sorting results

To resort the files in the legend click the left mouse button on one of the column buttons.

In case of equal values the last sorting criteria will be used. The last 5 sortings will be stored.

- **Curve** Color, style and name of the curve
- **Type** Type and format of the selected result file
- **Sub** Number of the result folder
- **Port** Number referring to the respective port
- **Ext** File extension if available in file name, e.g. **inc**
- **Path** Location of added files or folders
- **Desc** File descriptor or prefix if available in file name, e.g. **Directivity**
- **Freq** Frequency if available in file name
- **Phi** Phi angle if available in file name
- **Theta** Theta angle if available in file name
- **Parameter(s)** Sweep parameter if available in folder name
- **x,y,z** Position of a field probe

In case of parameter sweeps, optimizations, or multiport simulations the amount of result files can be very large and it is helpful to hide some of the entries. The automatic file detection can be limited by applying a file filter 25.1.6.

Column Menu entries (right click on column button):

- **Hide column** Hide selected column
- **Hide single value column** Hide all columns which have a single value
- **Show all columns** Display hidden columns
- **Edit Filter** Enter filter expression for a column

In order to find the corresponding file in the plot the files in the rows can be selected with a left mouse click. Multiple files can be selected by dragging the left mouse button or with **Ctrl** or **Shift** and left mouse click. After selection a menu can be opened with a right mouse click to hide or delete the files.

Row Menu entries (right click on row)

- **Show only and Autocolor** Show the selected file(s), recolor and hide all others
- **Show only** Show the selected file(s) and hide all others
- **Show** Display the selected hidden file(s)
- **Hide** Hide the selected file(s)
- **Delete** Delete the selected file(s) from the legend (Use **Update** to display again)
- **Show all** Show all files

- **Hide all** Hide all files
- **Assign Variable from Result**
- **Edit Style** Modify color and style of curve
- **Autocolor** Re-color selected files
- **Set Color** Select curve color
- **Set Linewidth** Select curve thickness
- **Set Linestyle** Solid, dashed or dotted line
- **Tree view** Alternative display of result files

Hint: Curve color table

By default up to 10 colors are used to display different curves. These colors are cycled with different line styles if more curves should be displayed. It is possible to modify and extend the number of colors by editing the color cycle file `user_defines.py` which is located in `$EMP/empire/kallisto/GUI/Q2D/`. RGB hexadecimal values, e.g. `'#ff0000'` for `'red'` are supported in the color list.

25.1.6. 2D Curve Display Setup - Filter

Filtering curves can be done with the **Filter Settings** button .

- Filters can be added and named. At the top the filters can be activated or de-activated for the current project
- The **Curve Name Filter** applies to the result name as listed in the **Curve** column, e.g. `s?_1`
- Other columns of the Curve Display Setup can be filtered by checkmarking the respective rows in the filter editor
- if `<All>` is selected no filter applies to the respective column. With `<Invert>` the checkmarked items are removed from the list. `<Empty>` displays all curves with no entry in this column.
- The unnamed row can be used to enter a filter for a matching expression, see Fig.25.3:
  - **Column Name**, e.g. entry: `Excite` in column **Port** yields the reflection parameters for S-Parameters
  - **Wildcards**, for one `?` or multiple `*` characters can be used to reduce the choice
  - **Expressions**, e.g. `>`, or `<` can be used to limit the value range

25.1.7. 2D Curve Display Setup - Marker

Markers can be created with the marker buttons ,  or , and are listed in the **Marker** Tab.

Right click on the marker opens the context menu and the marker can be controlled

- **Show** Enable marker
- **Hide** Disable marker
- **Remove** Delete Marker
- **Increment** Scroll marker to the right (also with right arrow key)
- **Decrement** Scroll marker to the left (also with left arrow key)

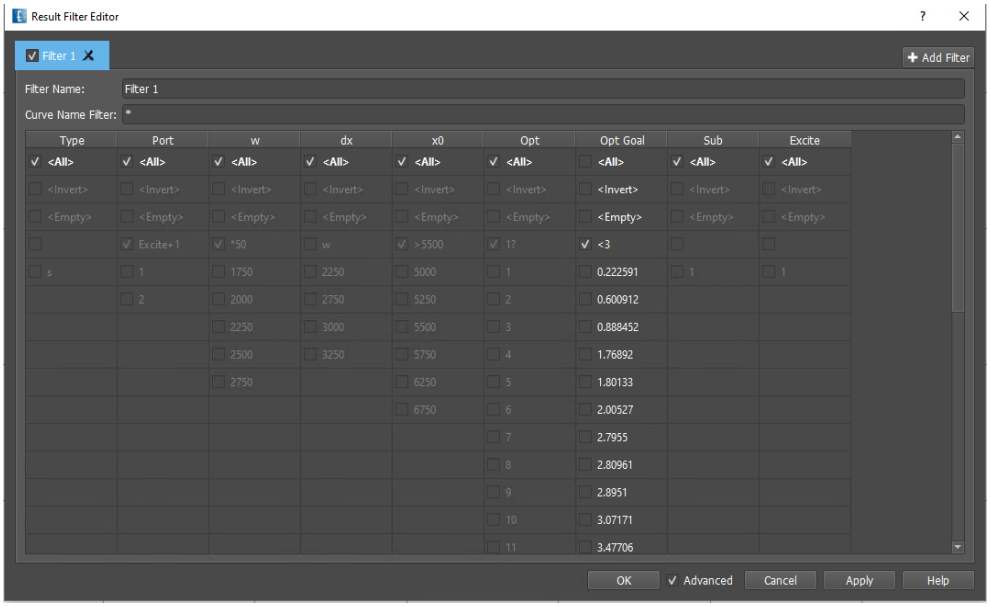


Figure 25.3.: Result Filter Editor Example

- **Clear Delta Marker** Delete Delta entry in table
- **Delta Marker** Calculate difference to selected other marker
- **Show all** Enable all markers
- **Hide all** Disable all markers
- **Remove all** Delete all markers
- **Auto Resize Columns** Extend all columns

25.1.8. 2D Curve Display Setup - Masks

Masks can be created and are listed in the **Masks** Tab.

Right click opens the context menu and the masks can be controlled

- **Add Mask** Create a new mask and open the mask editor, see Figure 25.4
- **Import Mask** Import a mask which has been exported i.e. in another project
- **Edit Mask** Open the mask editor
- **Delete Mask** Remove mask
- **Export Mask** Save mask for import into another project
- **Clone Mask** Copy mask for editing

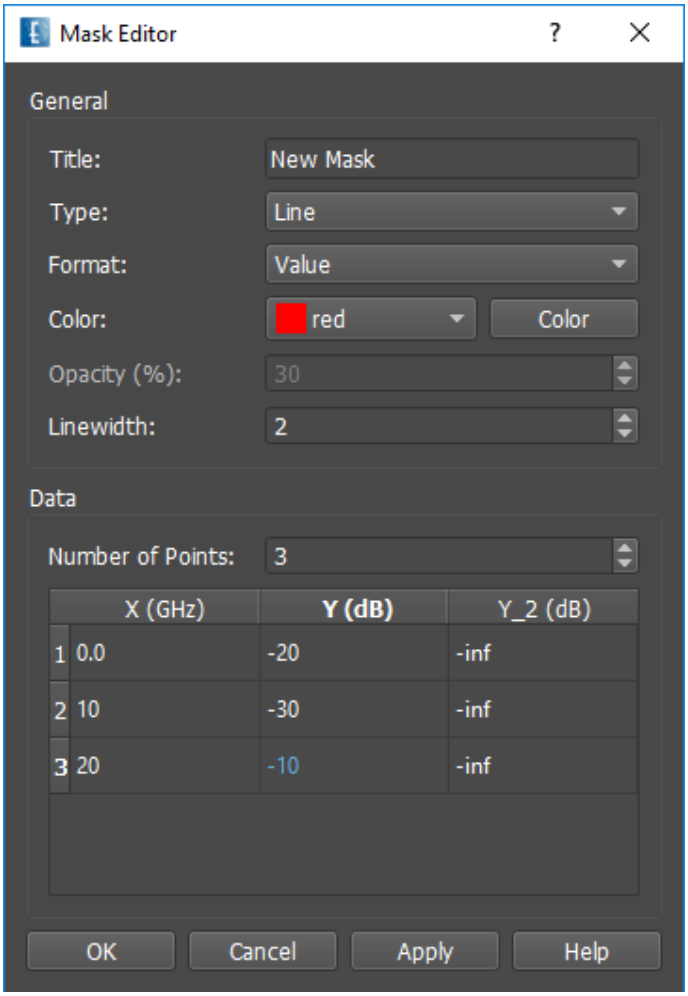


Figure 25.4.: Mask Editor

25.1.9. Variable Sweep & Curve Setup

By adding a sweep result curves can be plotted with variables as independent argument. In addition to **Plot Type** and **Plot Format** as described above the following items can be set up.

- **Variable** Name of variable as defined in **Design** Tab and used in Sweep or Optimization
- **Data** Magnitude, Phase, Real or Imaginary parts
- **Target** Determines the value of argument
  - **Value at Argument** Plot Type over Variable at an Argument. The argument can be specified with (e.g. S-parameter over variable at center frequency)
  - **Value at Minimum** Plot Type over Variable at Argument of Minimum. The argument of first minimum is used (e.g. S-parameter over variable at resonance frequency)
  - **Value at Maximum** Plot Type over Variable at Argument of Maximum. The argument of first minimum is used (e.g. S-parameter over variable at resonance frequency)
  - **Argument at Minimum** Argument over Variable at Minimum. The argument of first minimum is used (e.g. Resonance Frequency over variable)
  - **Argument at Maximum** Argument over Variable at Maximum. The argument of first maximum is used (e.g. Resonance Frequency over variable)

25.2. 3D Result Display

25.2.1. Near field

Near field results can be displayed together with the structure in the **3D Results Tab** provided that an **EM Field** monitor has been defined.

After simulation the results for each monitor are stored into files, e.g. `emvolume.dbx` for the near field recording in binary format. This file can be selected for animation. The displayed results can also be extracted into ASCII format with right click on the Field Monitor and select **Export Raw Data (full)**.

Refer to Chapter 19.1 for display option details.

25.2.2. Far field

Far field results can be displayed together with the structure in the **3D Results Tab** provided that an EM Far field Monitor has been defined.

After simulation the Huygen's box recording is stored into a file, called e.g. `Farfield_1` (binary format) which can be further processed to far field results with a far field transformation. After transformation in the postprocessing the results are stored in ASCII files `Farfield_1_fxx_pyy_tzz_ecomp` where `xx` refers to the frequency, `yy` to the phi angle, `zz` to the theta angle and `comp` to the polarization. These files can be selected for animation.

Refer to Chapter 19.3 for display option details.

In addition to the 2D curve display of far field cuts and 3D Display of radiation pattern a color map of the far field can be created with the aid of the **Plot Type = Far Fields (3D)** in the **2D Results** tab.

It displays a color map on a polar or rectangular plot.

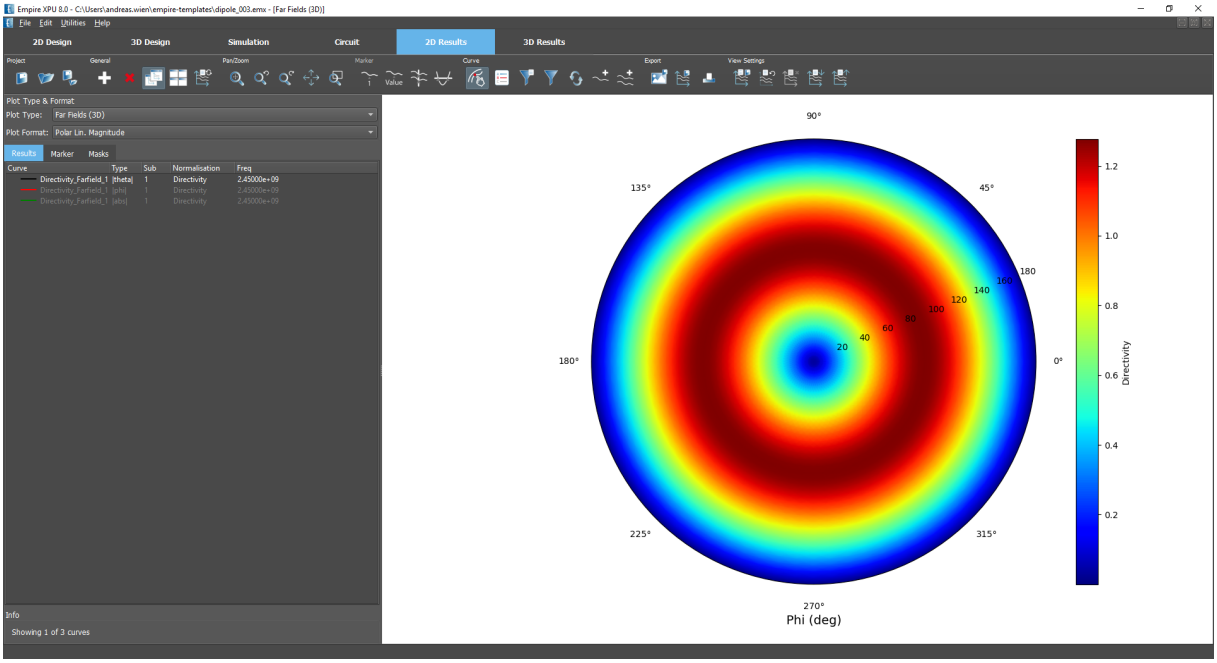


Figure 25.5.: Color map of a far field

25.3. Result Files

25.3.1. File location

Sequential excitation

For each excited port with unique port number `k` a result folder `sub-k` is created and the results of this simulation are stored therein. Some results are derived from several simulations, e.g. an S-Matrix, or a superposition, which will be stored in some higher level folders, e.g. `spar, ypar, superposition, mimo`

Simultaneous excitation

If all ports are excitated at once, only one result folder will be created. The folder number is derived from the lowest port number.

Plane Wave excitation

The result folder name is **sub-1** by default. A number may be chosen in the Plane Wave **Parameters - Options**

In case of parameter variations or optimizations, folders will be generated with the designated parameter combination, e.g. `v-opt-0001-a=xx-b=yy-c=zz` and similar result folders are generated in each variation folder.

25.3.2. File processing

Simulation results

During the simulation the following result files will be generated:

File name	Meaning	Occasion
et	Excitation signal	Always
utk	Voltage at Port k	Port k defined
itk	Current at Port k	Port k defined
wgport_j.dbx	Waveguide mode	TE-TM Waveguide Port j
emline_n.dbx	Line near field recording	EM Field Line Monitor n
emplane_n.dbx	Planar near field recording	EM Field Plane Monitor n
emvolume_n.dbx	Volume near field recording	EM Field Volume Monitor n
Farfield_n	Boundary near field recording	EM Farfield Monitor n

Here, the number n refers to the number of the respective field monitor. The value is automatically set and connot be changed by the user.

Hint: Output synchronization

Intermediates of the field dumps can be obtained by pressing the **Sync Output** button. An intermediate postprocessing is needed for displaying the near or far field due to normalization or tranformation purposes respectfully.

Postprocessing default results

By default, after the simulation the following result files will be generated:

File name	Meaning	Occasion
ef	DFT of Excitation signal	EM Simulaiton
ufk	DFT of voltage at Port k	Port k defined
ifk	DFT of current at Port k	Port k defined
Zk	Impedance at Port k	Port k defined
Farfield_n_fxx_p0_ecomp	Far field pattern in xz plane	Farfield Setup 1
Farfield_n_fxx_p90_ecomp	Far field pattern in yz plane	Farfield Setup 2
Farfield_n_fxx_ecomp	3D Far field pattern	Farfield Setup 3
Farfield_n_fxx_ecomp.log	Far field statistics	Farfield Monitor
Farfield_n_Rad_Eff	Radiation Efficiency vs frequency	Farfield Monitor
th.dat	Temperature distribution	Thermal solver

where fxx denotes the selected frequency in the far field definition and ecomp is the farfield component eabs, etheta and ephi.

Postprocessing results with excited port

In addition, if port j is excited:

File name	Meaning	Occasion
utj.inc (itj.inc)	Voltage (Current) of incident wave at Port j	
utj.ref (itj.inc)	Voltage (Current) of reflected wave at Port j	
ufj.inc (ifj.inc)	DFT of incident voltage (current) at Port j	
ufj.ref (ifj.ref)	DFT of reflected voltage (current) at Port j	
sk_j	Scattering parameter	Port k defined
Zj.in	Input impedance seen at port j	
Pj	Power at port j	

25.3.3. File formats

The following result files are stored in ASCII format.

Domain	Examples	Format	Comment
Time	et, ut1, it1,...	value [V, A]	at sampled timestep NSxDT (in Makefile)
Frequency	ef, uf1, if1, uf1.ref, ...	Frequency [Hz] Real Imaginary	DFT of TD result
Frequency	Z1, s1_1, P1,...	Frequency [Hz] Magnitude Phase [deg]	Processed from FD result
Angular 2D	Farfield_n_fxx_tyy_ezz	Phi [deg] Magnitude Phase [deg]	Theta sweep
Angular 2D	Farfield_n_fxx_pyy_ezz	Theta [deg] Magnitude Phase [deg]	Phi sweep
Angular 3D	Farfield_n_fxx_ezz	Theta [deg] Phi [deg] Magnitude Phase [deg]	Sweep Both
Space 3D	emvolume_n.dat	x y z [m] Components (Magnitude Phase [deg])	Monitor - Export Raw
Space 2D	e, h	x y z [m] Magnitude Phase [deg]	Field Line Monitor
Modal nD	name.snp	Frequency (S(1...n)_(1...n) Magnitude Phase [deg])	Touchstone file

Here fxx denotes a result at frequency xx, tyy and pyy stand for angle theta or phi at fixed angle yy and ezz is the selected polarization of the far field result.

26. Options

26.1. Editor Option Menu

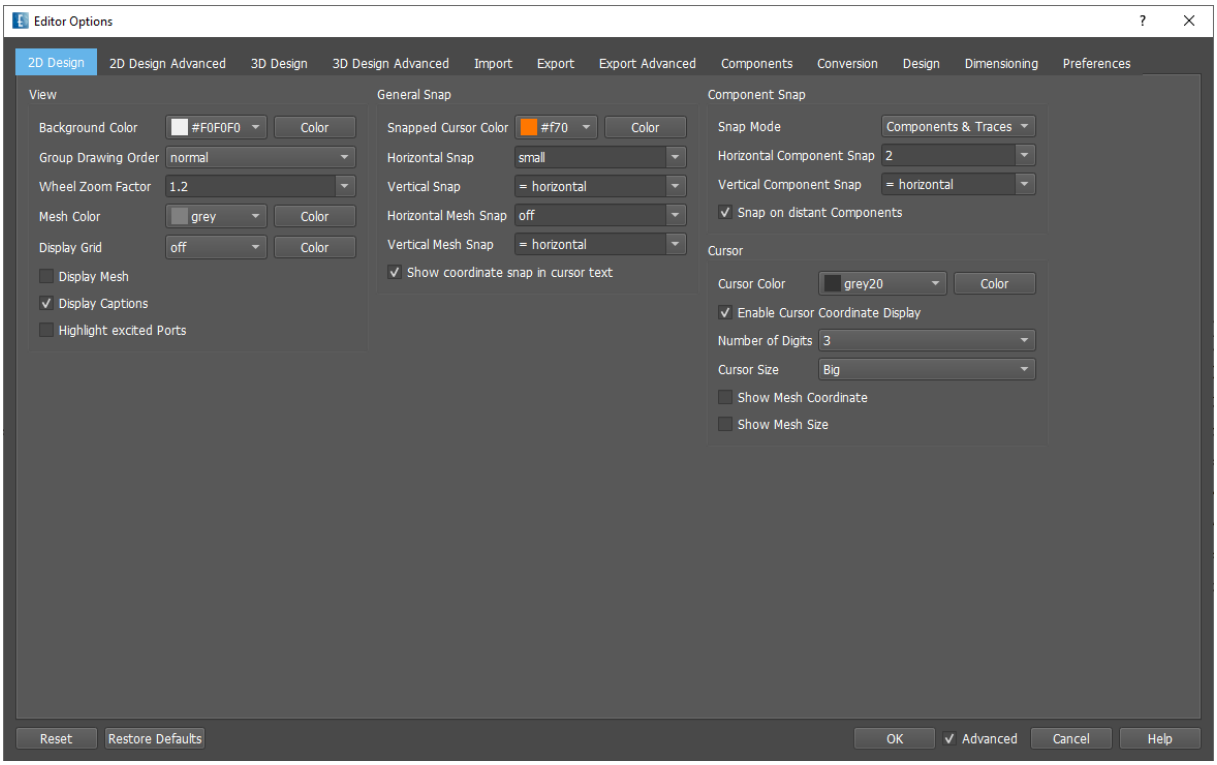


Figure 26.1.: Option menu

Some set-ups of the GUI, like grid display, defaults, or snap options can be saved within the options menu which is depicted in Figure 26.1 by clicking the button in **2D Design** or **3D Design** Tab. It is also possible to

- replace all changes with default values **Utilities - Default Editor Options - Restore**
- replace changes with default values in the current Tab by clicking **Restore Defaults**
- reset changes with values before opening the current Tab by clicking **Reset**

26.1.1. 2D Design

View

- **Background Color** is the background of the **2D Design** mode

- **Group Drawing order** If the tt 2D Design mode is set to **outlined view** objects can be drawn in reverse order if they lay over each other
- **Wheel Zoom Factor** Determines the velocity of the zoom when using the mouse wheel
- **Mesh color** Color of mesh
- **Display Grid** Display an equidistant grid similar to 3D Design mode

☐ **Display Mesh** By default the mesh lines are displayed at right and bottom bars. They can be displayed in the complete drawing area by checkmarking

☐ **Display Captions** By default port labels are displayed at port location. This can be de-activated for 2D Design here.

☐ **Highlight excited Ports** By default port excitations are marked with letter **E** after port number. Excited ports can be highlighted with green background by applying this switch.

General Snap

By using mouse input it can be very useful that only certain co-ordinate values can be accessed.

- **Snapped Cursor Color** If the cursor is snapped the cursor lines are highlighted with this color.
- **Horizontal / Vertical Snap** The snap interval can be set to constant values, e.g. 10 units. A smarter snap mode, like `big, medium, small` considers the current zoom factor of the drawing.
- The **Horizontal / Vertical Mesh Snap** sets the minimum distance in pixel to snap a mesh line. The value in the cross-hair cursor is then highlighted.

☐ **Show coordinate snap in cursor text** If eneabled, the snap resolution is displayed next to the cursor coordinates at the bottom of 2D Design.

Component Snap

- **Snap Mode:**
  - **Off** Disable snap on components
  - **Components** The cursor snaps to (lines of) components and draws symbols at the components edges

    × symbol if cursor is snapped on edge

    ☐ symbol if cursor is snapped to a corner

    ◊ symbol if cursor is snapped to a midpoint of 2 corners

    ▫ symbol if cursor is snapped to a center (circle, box)
  - **Components & Traces** If the cursor is moved to an object corner (or middle of 2 corners) traces (dashed grey lines) are drawn which are tangential (or normal) if the cursor is moved along the trace. Use the **Escape** key to stop drawing traces.
- **Horizontal / Vertical Component Snap** The selection accuracy can be chosen in pixel (objects within this distance can be selected).

☐ **Snap on distant components** If set to **on** the cursor snaps to objects edges which are away from the current position

Cursor

- **Cursor Color** Color of cross-hair cursor and coordinate numbers
- ☐ **Enable Cursor Coordinate Display** If disabled, coordinates are only shown at the bottom
- **Nnumber of digits** Number of digits displayed at cursor position
- **Cursor Size** Length of the cross hair cursor in pixel. **Big** covers the complete drawing area.
- ☐ **Show Mesh coordinate** If enabled, the current cell coordinates and numbers are displayed in addition to the coordinates
- ☐ **Show Mesh Size** If enabled, the current cell spacing is displayed in addition to the coordinates

26.1.2. 2D Design Advanced

Coordinate Snap

- **x** Automatic snap (depending on zoom factor) or snap in units in x-direction
- **y** Automatic snap (depending on zoom factor) or snap in units in y-direction
- **z** Automatic snap (depending on zoom factor) or snap in units in z-direction

Point

- **Point color** Color of entered points
- **Point size** Cross hair size of the point in pixel
- **Point text** Point number and coordinates can be displayed

Arrow

- **Arrow color** Color of entered arrow
- **Arrow rectangle color** color of rectangle spanned by the arrow, can be switched off
- **Arrow text** Arrow number and coordinates can be displayed

26.1.3. 3D Design

General

- **Background Color** is the background of the **3D Design** mode
- **Pixel Tolerance** Accuracy of cursor in pixel
- **Snap Resolution** in screen percentage or unit. The grid resolution (26.1.4) should be a multiple of this value in order to snap on the grid.
- **Tripod** Size of the uvw Tripod relative to screen or in units
- **Min. Normal Angle for Wireframe** Display tolerance for edges
- **Max. Number of Handles** Handles are displayed after selection. This defines an upper limit to prevent long display times

Options

- **Display Quality** If graphic is slow (e.g. Remote Desktop operation) this option can help to accelerate

☐ **Simple and Fast Cursor** If unchecked, the cursor appearance will change

☐ **Orthogonal Projection** If unchecked, 3D views are displayed in vanishing point projection mode

☐ **Animated View Change** If unchecked, view changes are abrupt without continous transition

☐ **Display Captions** By default port labels are displayed at port location. This can be de-activated for 3D Design here.

Snap Switches

- **Vertex Snap**
  - ☐ **Prominent Vertices** if checked, cursor snaps on visible vertices (blue circle)
  - ☐ **Edge Midpoints** if checked, cursor snaps on edge midpoints (yellow circle)
  - ☐ **Plane Midpoints** if checked, cursor snaps on plane midpoints (yellow circle)
- **Curvature Snap**
  - ☐ **Circular Curves** if checked, cursor snaps on cylindrical curves and displays radius and angle (white circle)
  - ☐ **Spherical Curves** if checked, cursor snaps on spherical curves and displays radius and angle (white circle)
- **Object Snap**
  - ☐ **Sides** if checked, cursor snaps on visible edges (orange circle)
  - ☐ **Planes** if checked, cursor snaps on visible planes (green dot)

26.1.4. 3D Design Advanced

Grid

- **Max Grid Size** Number of grid lines displayed in 3D Design mode
- **Grid Resolution** Grid line spacing in screen percentage or units
- **Center Grid Color** Grid display (close to origin)
- **Center Grid Transparency** Grid display (close to origin)
- **Center Grid Line Width** Grid display (close to origin)
- **Distant Grid Color** Grid display (far apart from origin)
- **Distant Grid Transparency** Grid display (far apart from origin)
- **Distant Grid Line Width** Grid display (far apart from origin)

Curvature Snap

- **Snap Angle** Snap resolution on circles in degrees
- **Minimum Angle** Determines range for curvature detection
- **Maximum Angle** Determines range for curvature detection

Sphere Snap

- **Snap Angle** Snap resolution on spheres in degrees
- **North Pole** Orientation of spherical coordinates

Coordinate Snap

☐ **Force Snap on cartesian snap** Deactivate object snap if enabled

- **x** Off or snap in units in x-direction
- **y** Off or snap in units in y-direction
- **z** Off or snap in units in z-direction

New Objects

☐ **Place Port Terminal at First Click** If enabled the port terminal (load and excitation) of sources - e.g. MSL - will be placed at the first mouse click (as in previous versions of EMPIRE). If disabled (default) port terminals are placed at the second mouse click, so they can be easily attached to existing structures.

26.1.5. Import

2D Layout

- **Import Scale Factor** The imported geometry is scaled with this value
- **Arc Resolution** Resolution of arcs in degree

☐ **Accept Zero Area Objects** By default, objects with no cross section are disregarded

- **Circle Recognition Points** Minimum number of points for the automatic recognition of circles.
- **Circle Recognition Tolerance** t Relative deviation from circular shape. Examples:
  - $t = 1$: Any selected polygon with minimum number of points will be converted to circles
  - $t < 0.29$: Keep squares
  - $t < 0.14$: Keep hexagons
  - $t < 0.07$: Keep octagons
  - $t = 0$: Only perfect circles

Empire

- **Scale Factor** Scale factor for the import of native Empire files or objects
- ☐ **Auto Convert Unit Scale** If enabled imported objects are scaled automatically
- **Shift (x y z)** Origin shift of imported Empire file or object

Advanced

- **HPGL pen width** Option for HPGL files

2D Layout Formats

- **Gerber**
  - ☐ **Merge GERBER Layers** If set to on, all objects in one group which are connected are merged to a single polygon
  - **GERBER merge tolerance** Maximum allowed distance in units for merging connections
  - **GERBER Zero Aperture** defines the thickness of a zero aperture
  - **EXCELLON decimal digits** Number of digits to import Gerber format based files
- **GDSII** The below options refer to the units before import. They are useful to avoid the import of micro vias or replace them by larger vias.
  - **GDS VIA size** Size of the replacing vias
  - **GDS VIA distance** Distance between vias
  - **GDS min. Obj size** Disregarding of small objects such as micro vias
- **DXF**
  - **DXF Connection Tolerance** Maximum allowed distance in units for merging connections
  - **DXF General Tolerance** If the difference between two co-ordinate values are below this threshold they will be considered as identical.
  - ☐ **Ignore layer in DXF block** Option for nested DXF layers
- **ODB++**
  - ☐ **Import Unknown Layer** If enabled, all available layers will be imported
  - **Shift (x y z)** Placement of import

3D Solids

- **Facets (STL/OFF/Nastran)** Polygonal mesh Represenation
  - **Shift (x y z)** Default shift vector for Facet import
  - ☐ **Separate Disjoint Parts** Facet data files may contain several objects. If set to ”on” the objects are split and placed in separate groups
- **BREP (e.g. Step/IGES/ACIS)** Boundary Represenation
  - **Wire Radius** Import B-REP wire bodies with radius in m
  - **Close small gaps** close gaps with the tolerance given in units
  - **Remove Sliver Faces** remove faces with high ratio longest to shortest edge
  - **Stitch Faces** Merge faces with merging tolerance entered in units
  - **Surface Tolerance** Acceptable deviation between original and imported data in units or fraction of wavelength
  - **Group depth** Select hierarchy level of imported data
  - **Group count warning threshold** Give warning if number of same level groups are too high

26.1.6. Export

2D Layout

Only Polygon-like objects can be exported as layout.

- **GERBER**
  - **GERBER Scale** Scale factor for the layout export
  - **GERBER Aperture Size (mm)** Holes in Gerber files are generated by apertures. The size of the aperture determines the accuracy of holes
  - **GERBER Float Format** Option for Gerber format
  - ☐ **Slice Polygons** avoiding holes if enabled
- **DXF**
  - **DXF Scale** Scale factor for the layout export
  - **Circle Tolerance** Deviation from circular shape in units
  - **Hole Treatment** defines how holes are exported into DXF files
- **GDS**
  - **GDS Export Unit in Meter** Because of the integer representation of the GDS format the value should be small enough

3D Solids

Solids are general 3D objects which are represented by surface triangles.

- **Facets (STL/OFF)** Polygonal mesh representation
  - **Export Unit** Target unit for the STL/OFF export format
  - ☐ **Separate Files by Group** If enabled all selected solids will be written into files defined by groups
  - ☐ **Separate Files by Components** If enabled all selected solids will be written into separate files
- **BREP (e.g. STEO/IGES/ACIS)** Boundary representation
  - **Export Engine** Adjust BREP engine:
    - **Open CASCADE** Based on Python OCC (boxes, cylinders, polygons)
    - **Primitive Planar** Export one planar facet per triangle
  - **Export Unit** Target unit for BREP export format
  - **Rasterize** Map solid coordinates to a grid with a spacing of this value

2D Image

- **Resolution factor** The resolution of the 2D image (2D Design Mode) can be enlarged by a factor, i.e. to create high resolution pictures for posters.

3D Image

The 3D image is the picture which is created in the Display mode (rendered objects including animations)

- **Resolution factor** The resolution of the 3D image can be enlarged by a factor, i.e. to create high resolution pictures for posters

☐ **Show coordinate system** Hide or display coordinate symbol (tripod) in 3D images

26.1.7. Export Advanced

Here, many plotting and machine control formats are available for layout export and their specific options can be set. The following formats can be exported with **File - Export - Advanced - 2D Layout** menu

- **HPGL**: Hewlett-Packard Graphics Language (printer control language)
- **UHT**: Format for processing equipment from UHT corporation. (uht_tools.py script required in working directory)
- **EXCELLON**: NC Drill file. The Excellon drill format is a subset of Gerber RS-274D and is used by the drilling and routing machines made by the Excellon corporation.
- **MLS**: Format for processing Laser machines from MLS GmbH

The following format can be exported with **File - Export - 2D Layout** menu

PDF:

Generate vector graphic from layout in 2D Design mode

- **PDF Page Size (mm)** Automatic or size in axb [mm]
- **PDF Layout Scale** Adjust line thickness and text size relative to layout size
- **PDF Layout shift** Adjust offset to page boundaries
- **PDF reference line width** absolute line width in Empire units
- **PDF fill style** wire frame (never) or as defined in group attributes
- **PDF Text Font** Print captions on or off, e.g port numbering
- **PDF color style** Black and white or as defined in group

26.1.8. Components

Polygons

- **Poly Rasterize** Value of the grid distance in units. If this function is applied to polygons their coordinates will be mapped to this grid
- **Poly Minimum Angle** Minimum angle in degrees used in function polygon simplification
- **Poly Tolerance** Value in units used for Polygon operations (Path, usertransformation, oversize, discretize)
- **Circle Resolution** Display resolution of a circle in degrees. Also used as resolution in Boolean operations
- **Rotpoly Subdivision** Resolution of a rotational polygon in number of segments

☐ **Automatic Circles** If enabled circle-like polygons will be converted to circles

- **Automatic Circle Tolerance** deviation in units

☐ **Line Poly Default Width** If enabled 1 cell is used for the line polygon after mapping on the grid (disabled = cell edges are used)

- **Poly Set Radius Option** Optional action for selected multiple points
  - **SET** Set entered radius to all selected points
  - **CW** Set entered radius for clockwise oriented corners
  - **CCW** Set entered radius for counter clockwise oriented corners
  - **SMOOTH** Apply smoothing alorithm for adjescent corners
- **Circle Recognition Points** Minimum number of points of a polygon for the function circle recognition
- **Circle Recognition Tolerance** t Relative deviation from circular shape. Examples:
  - $t = 1$: Any selected polygon with minimum number of points will be converted to circles
  - $t < 0.29$: Keep squares
  - $t < 0.14$: Keep hexagons
  - $t < 0.07$: Keep octagons
  - $t = 0$: Only perfect circles
- **Internal Line Angle** Angle of cutting line between hole(s) and polygon outside

- **Advanced**
  - ☐ **Rotpoly Fast Discretization** Meshing algorithm for rotational polygons
  - – **Show bounding box** Representation of the polygon in a side view

Bond Wires

Here, the default values for creating bond wires using the function ”Create Bondwire” can be set The angle is defined between transverse plane and wire

- **Slope Angle phi0** Angle at start point
- **Slope Angle phi1** Angle at end point
- **Number of Points in Bond Wire** Resolution of the wire
- **3D Display Linewidth** Thickness used for display

Via Fence

Adjustments for automatic generation of via fences for multi-layered circuits. Via fences can be generated for polygons only. The via fence is created for each side of the polygon if the side corners are selected. The vias are created in the center of the catch pad.

- **Geometry**
  - – **Distance to Poly** Distance in units between polygon and catch pad
  - – **Via Diameter** Diameter of the vias (should be smaller than pad width)
  - – **Via Distance** (Maximum) Distance between the vias
  - – **Pad Width** Width of the catch pad
- **Groups**
  - – **Via A Groups** Group name of the vias (if not available the group will be created)
  - – **Via B Groups** If name different to Via A group name vias alternated will be created (staggered vias)
  - – **Pad Groups** Group name of the catch pad
- **Logging** Window or File output

Generate Strip

- **Strip Width** default value for strip creation
- **Strip Mode** default mode for strip creation

Solids

- **Min. Normal Angle in 2D display** Wire frame representation of solids
- **Max Number of Solid Handles** Limit for the number of displayed handles when a solid is selected

☐ **Warn for improper objects** Warning messages for unclosed or self-intersecting solids

☐ **Save all Triangles in ASCII** By default solid are saved in binary format in Empire files. If enabled they will be saved in plain text.

- **Advanced**
  - **Relative Accuracy** used for solid operations
  - **Advanced - 2D Draft Display** Display acceleration by neglecting triangles
- **Refine**
  - **Length Target** Increase resolution (in units) of large faces
  - **Max # faces** Maximum number of faces used for refinement
- **Simplify**
  - **Angle Threshold** Ignore triangles with small angles
  - **Length Threshold** Ignore triangles with small sizes
  - **Solid Rasterize** When executed vertex coordinates will be rounded to multiples of entered value in units.

26.1.9. Conversion

Default options for conversion operations.

- **Convert to Polygon** This operation can be applied to Boxes or Solids.
  - **Cutting Plane** Either specific plane or dependent on the current view or grid
  - **Tolerance** Maximum error for object and plane intersection in units
  - **New height** of the resulting polygon

  ☐ **Keep object's groups** If unchecked, groups with zero objects will be deleted

  ☐ **Merge resulting polygons** If unchecked, all polygons are kept separated

  ☐ **Simplify resulting polygons** If checked, 180 degree corners are eliminated

  ☐ **Apply Circle Recognition** Convert polygon into circle if enabled

26.1.10. Design

- **Design Check Tolerance** Default value in units used for design rule checks
- **Rotation Angle** Default angle in degree used for rotation operations
- **Minimum Auto Thickness** Default thickness in units used for voltages or current boxes
- **Select Duplicate Object Tolerance** If 0 all objects with equal size will be selected
- **QTEM Port Drawing Mode** If set to **FAST** QTEM boundaries are only updated after manual redraw.

26.1.11. Dimensioning

- **Dimensioning Number of Digits** Number format
- **Dimensioning Scale Factor** Unit multiplication
- **Dimensioning Tic Size** Distance numbers to dimensioning line
- **Arrow Distance** Size of arrows
- **Arrow Heads** Shape of arrows
- **Font Size** Text size
- **3D Measure Distance** Distance between line and number in Display mode

26.1.12. Preferences

The preferences set here effect on the current input file, only.

- **Misc**
  - ☐ **Start in 2D Design** if enabled Empire starts in the 2D Design mode next time the file is opened.
  - ☐ **Create Backup Files** The automatic creation of backup files can be selected.
  - ☐ **Display Hints in Sidebar** If enabled hints will be displayed if available.
  - – **Last Recently Used Files** number of recently opened files listed in start-up window and in **File** menu
  - – **Icon Size** The default Icon size is 24x24 pixel. It can be enlarged to 32x32 or decreased to 16x16
  - ☐ **Enable Result Cache** Used for data in **2D Results** tab
  - – **Window Tile Mode** Choose project folder of file name only for heading
- **Logo**
  - – **Logo Display Mode** The logo is displayed in the 2D and 3D Design and 3D Results tabs at the specified location
  - ☐ **Display Logo in background** If enabled, objects are drawn in the foreground
  - – **Logo Size** choose from several predefined sizes
- **Scripting**
  - – **Current Working Directory**
    - ∗ **Script File Directory** Location of selected script file
    - ∗ **Keep** Location of current project folder
- **Boolean operations**
  - ☐ **Keep Boolean History** If enabled, boolean operations on solids will be recorded and executed only for simulation. This allows the use of parametric solids.
  - ☐ **Simplify Polys** If enabled, a simplification will be applied after executing a boolean operation with polygons
  - – **Poly Height Difference Handling** Select preferred outcome of operation:
    - ∗ **Ask Each Time** - Choose below options at occurrence
    - ∗ **2D Outcome** - Polygon with height of first selected object

    - **3D Outcome** - Solid, triangulated surface (FACET)
  - **Preferred 3D Outcome Type**
    - **Polygons** - multiple sliced polygons, keeping BREP property
    - **Solids/Surface** - single solid, triangulated surface (FACET)
  - **Engine of Boolean Operations** New operation engine (scad) introduced in version 7.60. Old engine (gts) available for compatibility reasons.

26.2. 2D Result Options

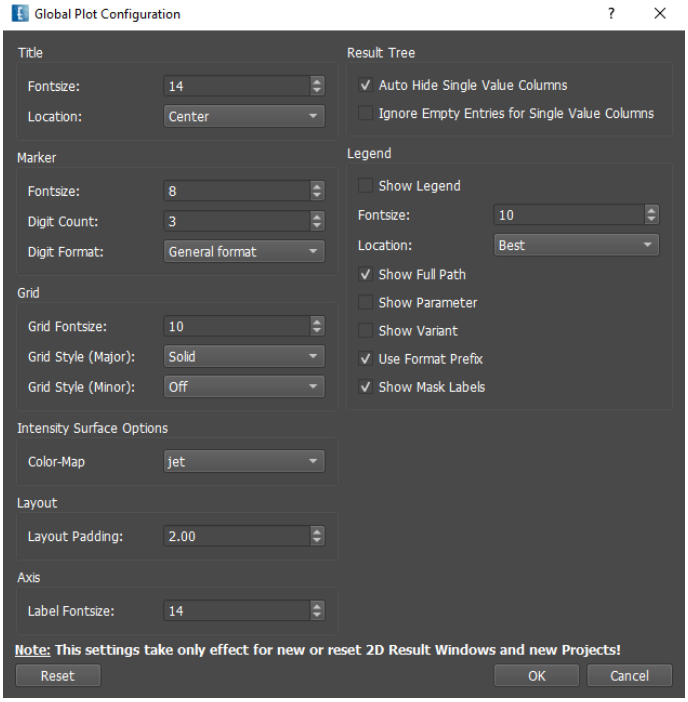


Figure 26.2.: Option menu for 2D Results

Global Plot Configuration for the 2D Results tab can be set by clicking the button when the **2D Results** tab is active. This setting takes only effect for new or reset 2D Results windows and new projects.

- **Title**

  Title appearance can be adjusted here

  - **Fontsize** Size of title
  - **Location** Position of title

- **Marker**

  Marker appearance can be adjusted here

  - **Fontsize** Size of marker label
  - **Digit Count** Number of digits in marker label

  - **Digit Format** Select number display, e.g. scientific format

- **Grid**

  Grid appearance can be adjusted here

  - **Grid Fontsize** Size of grid label
  - **Grid Style (Major)** Line style of major tics
  - **Grid Style (Minor)** Line style of minor tics

- **Intensity Surface Option**

  Set map for 2D color plots, e.g. **Far Fields (3D)**

  - **Color-Map** select a color gradient from list

- **Layout**

  Space between plot and frame can be adjusted here

  - **Layout Padding** Fraction of the grid fontsize

- **Axis**

  Axis label size can be adjusted here

  - **Label Fontsize** Size of the axis label

- **Result Tree**

  Number of columns of the result tree can be adjusted here

  -

  ☐ **Auto Hide Single Value Columns** Show only columns with different values

  ☐ **Ignore Empty Entries for Single Value Columns** Show only columns with values

- **Legend**

  ☐ **Show Legend** The legend can be enabled with a checkmark

  - **Fontsize** Size of legend characters
  - **Location** Position of legend in graph

  ☐ **Show Full Path** Location of result file relative to project folder

  ☐ **Show Parameter** Include variable names

  ☐ **Show Variant** Include variant names

  ☐ **Use Format Prefix** Include prefix if applicable

  ☐ **Show Mask Labels** Include mask if applicable

26.3. 3D Result Options

Options for the 3D Result tab can be set by clicking the  button when the **3D Results** tab is active.

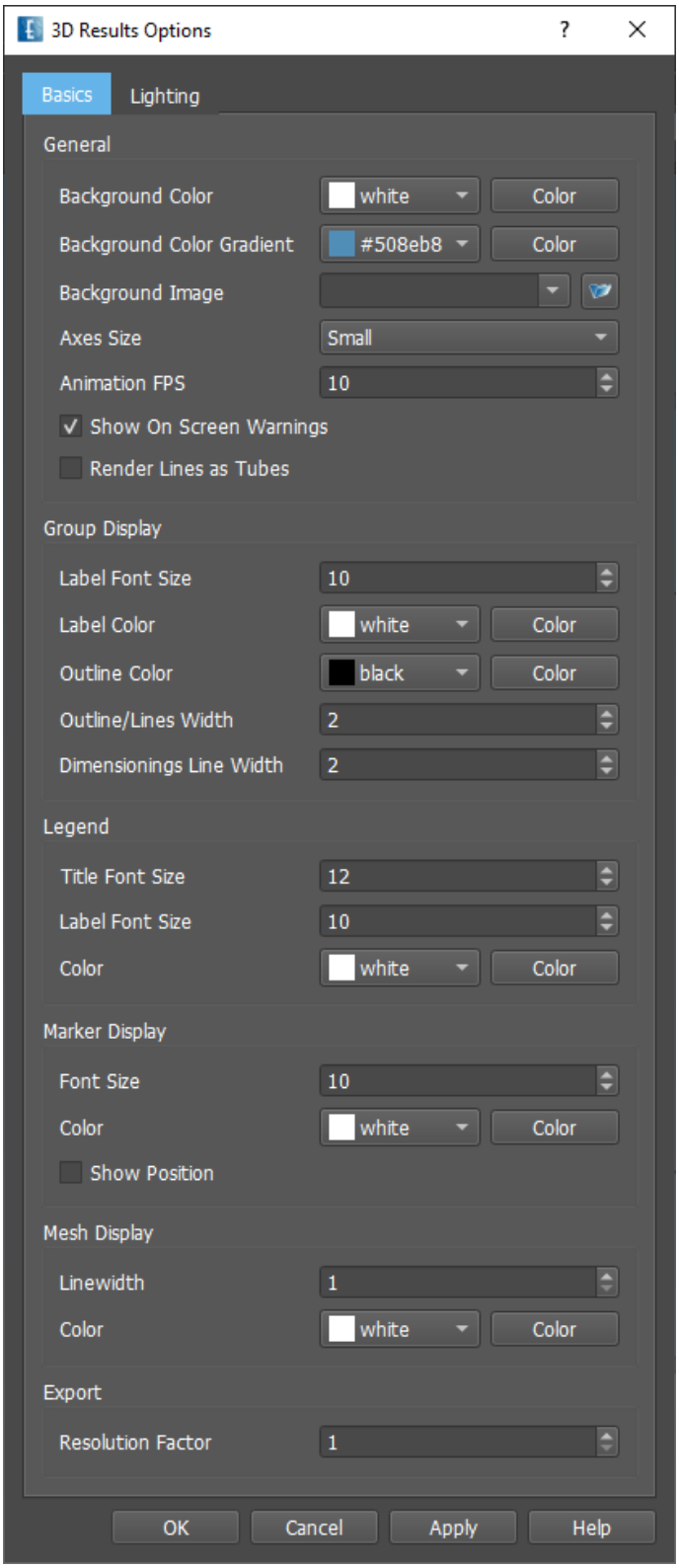


Figure 26.3.: Option menu for 3D Results

26.3.1. Basics

- **General**

  Background, animation velocity and Tripod appearance can be adjusted here.

  - **Background Color** Color used in the **3D Result** mode
  - **Background Gradient** Gradient transition of color in the **3D Result** mode
  - **Background Image** .png file to be used as background
  - **Axes Size** Size of tripod shown in image
  - **Animation FPS** Velocity of animations in frames per second
  - ☐ **Show On Screen Warnings** Show / hide messages in image
  - ☐ **Render Lines as Tubes** Display option for thin wires

- **Group Display**

  Port labeling and object display can be adjusted here.

  - **Label Font Size** Size of port label
  - **Label Color** Color of port label
  - **Outline Color** Color of wire frame of objects
  - **Outline/Lines Width** Width of wire frame of objects
  - **Dimensionings Line Width** Width of lines for dimensioning

- **Legend**

  Animation legend can be adjusted here

  - **Title Font Size** Size of legend title
  - **Label Font Size** Size of legend entries
  - **Color** Color of legend entries

- **Marker Display**

  Markers in animations can be created by right click on field display and select **Add Marker**. The position is displayed by a small arrow

  - **Font Size** Size of marker font
  - **Color** Color or markers and arrow
  - ☐ **Show Position** Display coordinates if enabled

- **Mesh Display**

  Mesh in **3D Results** tab can be enabled by clicking the **Mesh Visualization** button .

  - **Linewidth** Thickness of lines for mesh display
  - **Color** Color of lines for mesh display

- **Export**

  Adjustment for **Export image** 

  - **Resolution Factor** Image resolution enhancement (compared to screen pixels)

26.3.2. Preferences

Logo

- **Logo Display Mode** Define if and where the logo should be displayed in the canvas
- ☐ **Display Logo in the background** Choose drawing order
- **Logo size** Adjust size

26.3.3. Lighting

By default an ambient light is used to illuminate the scene. Its strength can be set by adjusting the sliders for **Geometry** or **Field Monitor**.

Several light sources (Head Light, Camera Light, Scene Light) can be added to illuminate the scene by right click in the **Lights** frame and select **Add Light**

- **H** Head Light with Intensity
- **C** Camera Light with Intensity and direction θ, φ (moves with camera)
- **S** Scene Light with Intensity and direction θ, φ (fixed to scene)

27. Project Wizard

EMPIRE XPU™ starts with a project wizard for an easy set up of a simulation structure. Tutorials are particularly developed for beginners to get started. Experienced users will also benefit from templates and examples, since many settings are done automatically and the structure can be modified according to the current needs.

Templates make use of built-in synthesis formulas in order to match impedances or resonance frequencies.

- **Common template feature**
  Every template comes with switches to record and display the near field if desired. Additionally, for antenna templates a far field computation can be applied. Some templates allow the usage of sweep variables.
- **General Settings**
  The drawing unit can be specified. The following drawing units can be used within EMPIRE XPU™: nm, um, mm, m, mil, inch, the default value is set to um. The synthesis formulas assume vacuum as background material.
- **Frequency**
  Here, the frequency range of interest can be specified. Some antenna templates as well as rectangular waveguide templates set the frequencies automatically according to their template parameters. If frequency range needs to be changed in this case, they can be modified later on in the simulation setup.
- **Loss calculation**
  The loss models for the narrow(5%), medium(50%) and broad band (95%), of the dielectric and the conductor can be entered here.

To select a **Template** press the associated plus sign in the **Selection** area, adjust **Parameters**, and press the button **Open Selection**.

27.1. New Project

This Tab is intended to start from scratch or to open some documentation (also available in the installation folder). In **General** parameters as described above can be set in advanced.

27.2. Open Project

This Tab lists recently used project files which can be reloaded for quick access. Project information is displayed on the right with a preview thumbnail image. If the project is not listed a project can be browsed by using the top button **Load Project**.

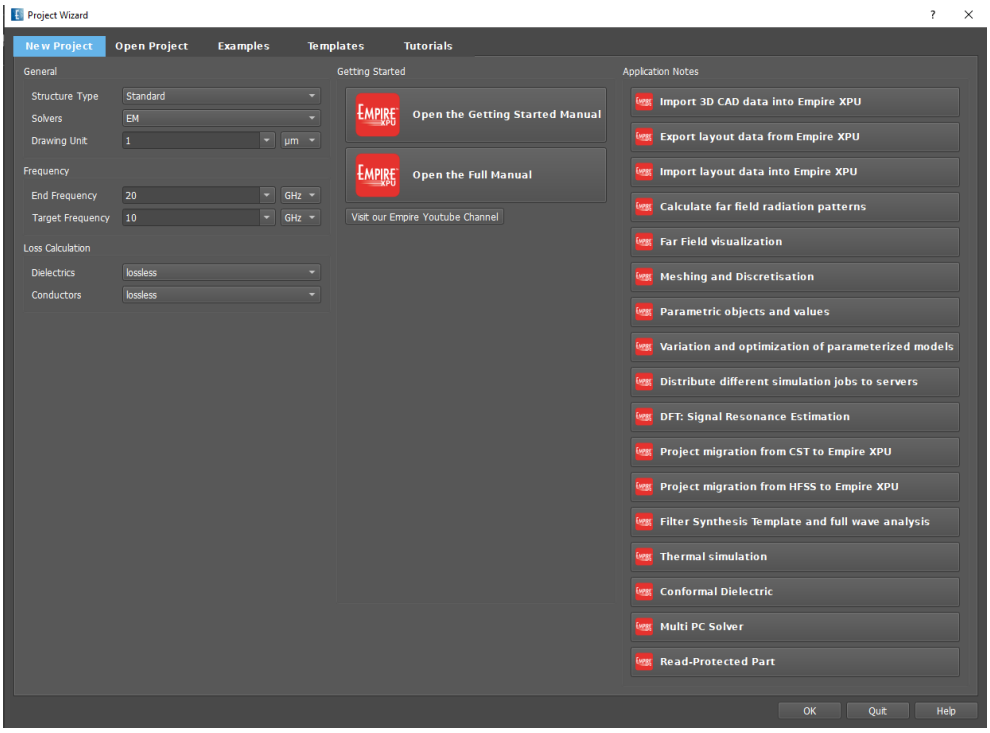


Figure 27.1.: Project Wizard: New Project

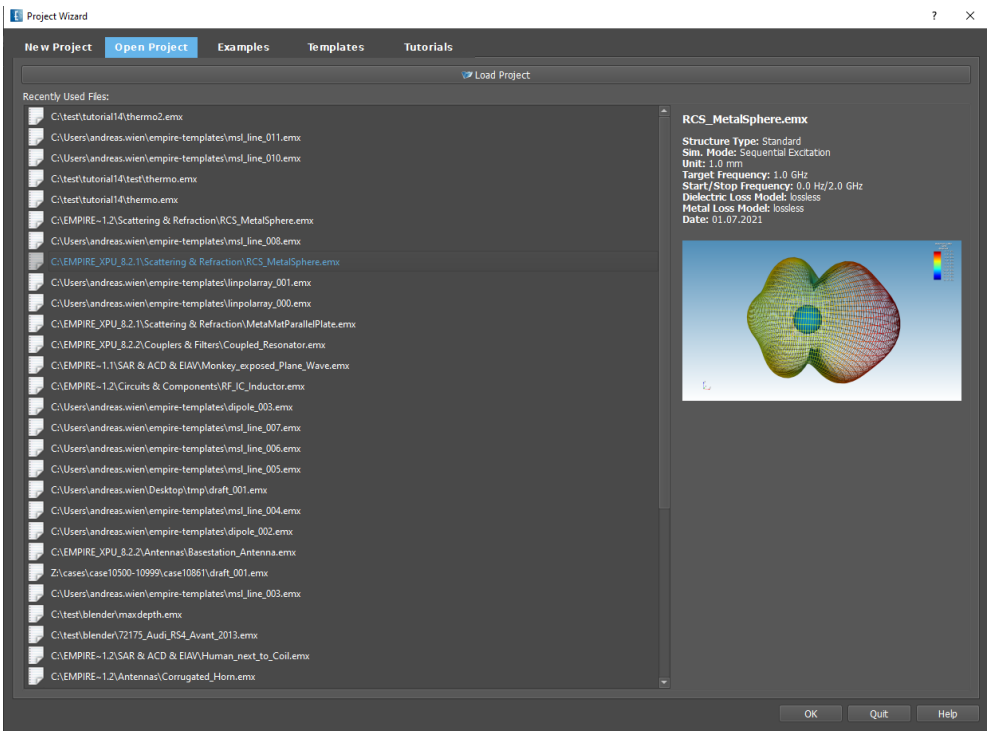


Figure 27.2.: Project Wizard: Open Project

27.3. Examples

Here, many examples with fixed parameters can be previewed and selected from a list. They can also be found in the EMPIRE XPU™ installation folder. Their results are already calculated and can be viewed in the **2D Results** or animated in the **3D Results** tabs without simulation.

Examples are stored in the installation folder if selected during the installation process.

27.3.1. Example quick reference guide

This guide also serves as a function test for the graphics.

1. Start Empire from Desktop
2. Select Examples
3. Choose Basestation Antenna Example. There are groups for dipoles, ports and a reflector. Drag right button to rotate the model.
4. Select 2D Results to see input reflection of the antenna
5. Select 3D Results to see radiation Pattern and near field. Drag right button to rotate the model for a perspective view.
6. Click Start Animation button  to see the animated nearfield

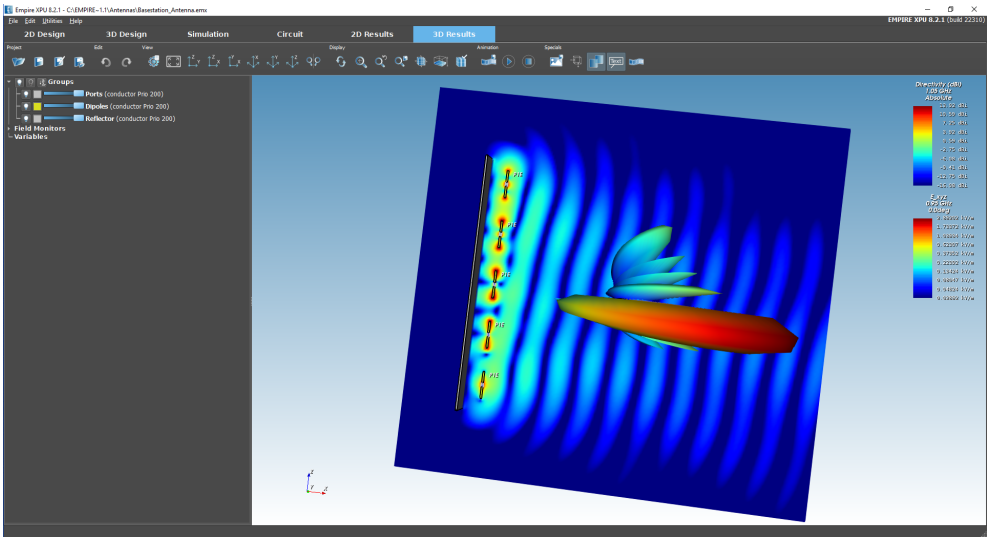


Figure 27.3.: Base station example

27.4. Templates - Transmission Lines & Waveguides

These templates generate very basic transmission line structures such as microstrip, stripline, coplanar, coaxial, rectangular and parallel plate wave guides.

27.4.1. Template quick reference guide

This guide also serves as a function test for the simulation flow.

1. Start Empire from Desktop
2. Select Templates
3. Choose Filters - Microstrip Tapped Hairpin
4. Click **Start Simulation** - Confirm messages. Now the structure will be simulated which takes some seconds
5. When State displays **Finished** select 2D results to see S-Parameters
6. Switch to 3D Results and open **Field Monitors** on the left, switch on light bulb
7. In **Groups** switch off bulb to see near field
8. Click **Start Animation** to see the near field at desgin frequency

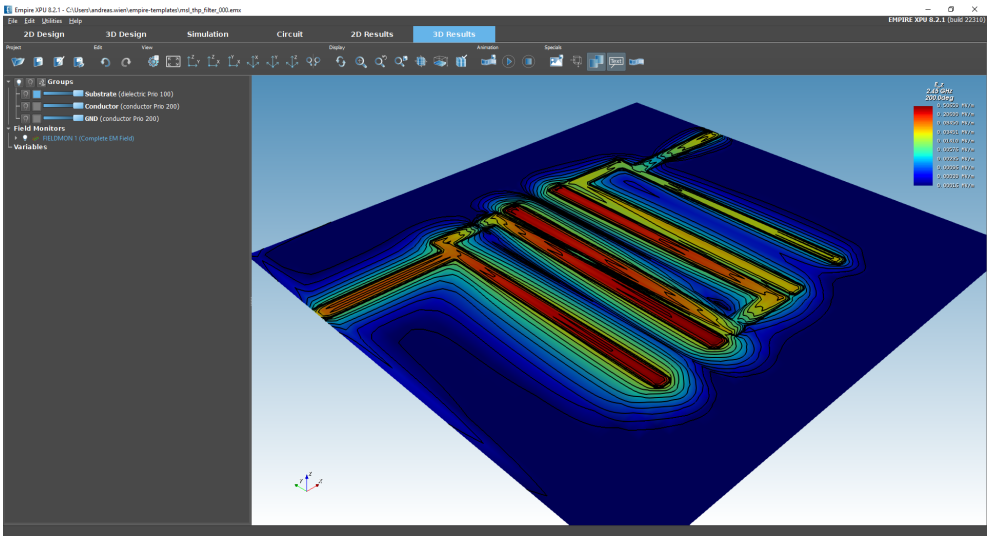


Figure 27.4.: Filter Template

27.4.2. Microstrip Line

This template sets up one ground planes, a substrate with editable height and a signal line on top. For this template three different setups can be chosen: impedance, line, and SMD. Using the switch **Calculate Width From Impedance Formula**, either the line width or the characteristic line impedance can be specified.

Impedance

The transmission line is terminated with an absorbing port and absorbing boundary condition at one side. Using this absorbing port the line is effectively terminated with its characteristic line impedance. So it can be used to calculate the line impedance which is available as result Z2. The reference impedance Z1 is derived from the QTEM port eigenmode analysis.

Line

The **line** is terminated with QTEM (Quasi TEM) concentrated ports at both sides. The reference impedance Z1 is derived from the QTEM port eigenmode analysis.

SMD

These templates set up SMD chip components (Resistor, Inductor, Capacitor) mounted on a microstrip line. The SMD size along with its value is placed with its pad between two microstrip line ports. The SMD components are modeled in the sense that they consist of metal caps and an inner sheet with the defined electric properties. This allows an accurate modelling taking into account parasitic effects of the pads and caps of the SMD.

The **SMD Port** template can be used to model an SMD component, using a circuit simulator. Between the two metal caps of the SMD a lumped port is placed. By exporting the s-matrix of the 3 port structure (two microstrip ports and one inner SMD port), external SMD data can be processed using a circuit simulator.

27.4.3. Strip Line

This template sets up two ground planes, a substrate with 2 times the height and a signal line in the center. For this template two different setups can be chosen: impedance and line. Using the switch **Calculate Width From Impedance Formula**, either the line width or the characteristic line impedance can be specified.

Impedance

The transmission line is terminated with an absorbing port and absorbing boundary condition at one side. Using this absorbing port the line is effectively terminated with its characteristic line impedance. So it can be used to calculate the line impedance which is available as result Z2. The reference impedance Z1 is derived from the QTEM port eigenmode analysis.

Line

The **line** is terminated with QTEM (Quasi TEM) concentrated ports at both sides. The reference impedance Z1 is derived from the QTEM port eigenmode analysis.

27.4.4. Coplanar Waveguide

This template sets up a substrate with editable height and a signal line and two ground planes on the top. For this template two different setups can be chosen: impedance and line. Using the switch **Width Calculation From Impedance Formula**, either the line width or the characteristic line impedance can be specified.

Impedance

The transmission line is terminated with an absorbing port and absorbing boundary condition at one side. Using this absorbing port the line is effectively terminated with its characteristic line impedance. So it can be used to calculate the line impedance which is available as result Z2. The reference impedance Z1 is derived from the QTEM port eigenmode analysis.

Line

The **line** is terminated with QTEM (Quasi TEM) concentrated ports at both sides. The reference impedance Z1 is derived from the QTEM port eigenmode analysis.

27.4.5. Coaxial Waveguide

This template sets up an inner and outer conductor and a dielectric material between them. Using the switch **Calculate Dielectric From Impedance Formula**, either the dielectric material or the characteristic line impedance can be specified.

Impedance

The transmission line is terminated with an absorbing port and absorbing boundary condition at one side. Using this absorbing port the line is effectively terminated with its characteristic line impedance. So it can be used to calculate the line impedance which is available as result Z2. The reference impedance and load at exciting port is 50 Ohm by default.

27.4.6. Rectangular Wave Guides

Rectangular Wave Guides (WG) are easily set up using this template. Here the common WG types and their frequency range are listed. The structure uses electric wall at the 4 sides of the WG and absorbing walls in the direction of propagation. Two waveguide ports to operate in the first TE Mode at either end of the wave guide are defined. The frequency range is set from $0.8 \times f_c$ (cut-off frequency) to the max operating frequency of the wave guide that is also given in the selection list. WG devices like filters, couplers or diplexers can be easily generated by modifying this structure.

27.4.7. Plate Plane Wave

With this template a parallel plate waveguide is modeled which is equivalent to a plane wave traveling normal to the waveguide cross section. Using this configuration it is possible to model a plane wave and evaluating transmission and reflection characteristics or model a unit cell of an EBG structure.

27.4.8. Conformal

This template sets up a bent substrate with editable thickness and bump height. If the switch **Variable Substrate Thickness**, is disabled the normal distance between top and bottom is maintained. If enabled the distance in z-direction is kept constant. Using the switch **Calculation Width From Impedance Formula**, either the line width or the characteristic line impedance can be specified. The reference impedance Z1 is derived from the QTEM port eigenmode analysis.

Conformal Microstrip Line

This template sets up the bent substrate, a signal line on top and a gound planes at bottom.

Conformal Coplanar Waveguide Line

This template sets up the bent substrate, a signal line and two ground planes on the top.

27.5. Templates - Coupled Transmission Lines

For both microstrip and stripline two coupled lines can be configured and the odd or even mode can be selected.

27.6. Templates - SMD chip components

These templates helps in modelling single SMD chip components (Resistor, Inductor, Capacitor) on a substrate. The SMD size along with its value is placed with its pad between two lumped ports. The SMD components are modeled in the sense that they consist of metal caps and an inner sheet with the defined electric properties. This allows an accurate modelling taking into account parasitic effects of the pads and caps of the SMD.

27.7. Templates - RLC

27.7.1. Rectangular Inductor

With the number of turns n, the line width and the slot width a rectangular inductor is placed on top of the substrate material. Two lumped ports at the inner as well as the outer line end are defined. By replacing the inner short, a stub conductor to ground is obtained and the inductance can be found from the impedance slope. Using the loss models of the dielectric as well as of the conductor, the quality factor of the coil can be computed. The simulation box is confined by magnetic walls except for the z_min direction which uses an electric wall. In order not to strongly effect the near field and the simulated inductance the wall should be placed at a distance of one coil diameter away from the structure. The simulation bandwidth is computed automatically taking into account the total length of the structure.

27.7.2. Interdigital Capacitor

By specifying finger number and dimensions a capacitor is placed on top of a substrate and ports are defined at the pads. Depending on the chosen size the end frequency will be adjusted in order to avoid box modes.

27.7.3. Meander Resistor

By specifying meander dimensions a resistive line is placed on top of a substrate and ports are defined at the pads. Depending on the chosen size the end frequency will be adjusted in order to avoid box modes.

27.8. Templates - Antennas

27.8.1. Dipole

The dipole antenna can be specified either by its resonance frequency or its physical length. The dipole is excited by a lumped port in the center. The near & far field calculation are performed at the resonance frequency of the antenna. The thickness of the dipole may not be smaller than 1e-3 of its length to ensure a quick simulation time. This is a very basic example to compute an antenna in free space. It can also be useful for experienced users to model their individual antennas and just use the pre-settings (Far field computation, Simulation settings and Simulation box) for the structures and frequencies of their interest.

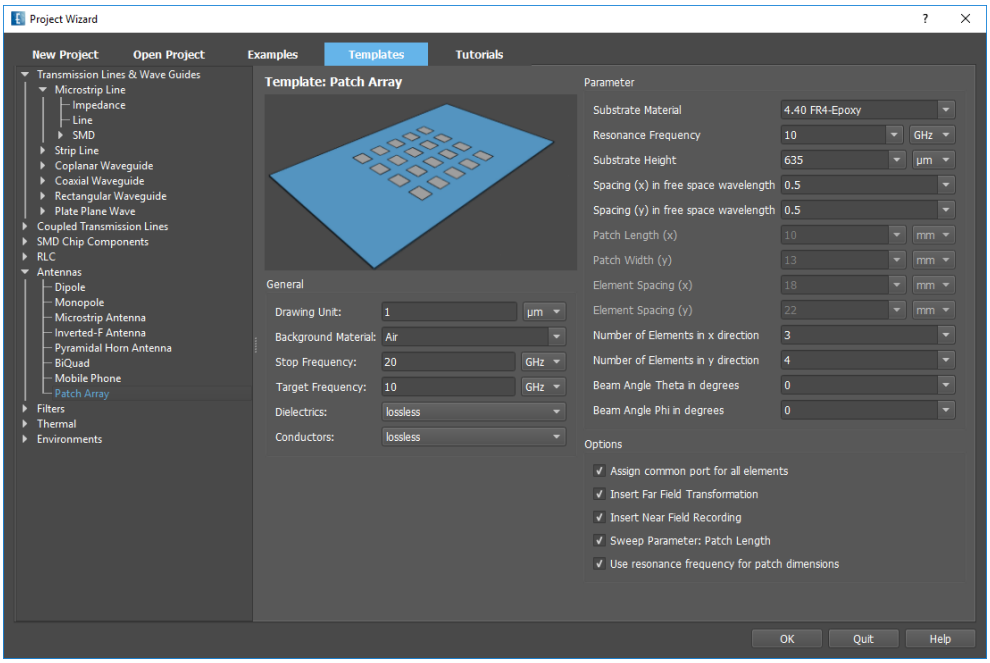


Figure 27.5.: Patch array template

27.8.2. Monopole

The monopole antenna basically has the same setting as the dipole antenna. It is resonating at a quarter wavelength. Additionally, a finite ground plane can be employed in the model with the dimension to be specified in the template. This model can be widely used for other types of antennas by specifying the ground plane size and replacing the monopole arm with individual radiating elements.

27.8.3. Microstrip Antenna

This template will generate a very basic patch antenna on an infinite ground plane. The substrate material permittivity (default Rogers Duroid 5880) as well as the height h (default 1524 um) are given in the initial setting. The $\frac{\lambda}{2}$ operating frequency needs to be specified. The patch length and the size of the total simulation box are calculated based on these parameters. The switches **Insert Far Field transformation** and **Insert Near field recording** allow to access the near field data and the far field data at the resonance frequency. After running the simulation the monitors for the near and far field animation are activated to see the fields in the 3D mode.

27.8.4. Inverted-F Antenna

This template will generate a basic IF antenna on a substrate material (default Rogers TMM-10) with the height h (default 508 um) given in the initial setting. The operating frequency needs to be specified. The IF length and the size of the total simulation box are calculated based on these parameters. The two switches **Insert Far Field transformation** and **Insert field dump** allows to access the near field data and the far field data at the resonance frequency. After running the simulation the field monitors for the near and far field animation are activated to see the fields in the 3D mode.

27.8.5. Pyramidal Horn Antenna

This template will generate a pyramidal horn antenna with a recangular waveguide feed. The operating frequency is determined from the selected waveguide. The opening angles, wall thickness and horn length can be selected. The switch **Insert Near field recording** allows to access the near field data at the target frequency. A far field monitor is defined at the operating frequency.

27.8.6. Mobile phone

A generic model of a cellular phone is implemented to model multi band antennas in mobile devices. The Antenna height is the distance between the upper antenna structure and the printed circuit board (PCB) and is set to 6mm by default. With the two geometric parameters **Board length** (default 107mm) and **Board width** (40 mm), the total size of the device can be adapted to the dimension of interest. This template is particularly useful to setup a quick simulation for any kind of mobile devices. The position, shape and size of the Antenna structure can be rearranged and modified in order to tackle the customer's needs. The frequency range for this template is automatically set from 500 MHz to 2.5 GHz. And the Farfield computation is done for 920 MHz. For this template the dimensions of the cell phone should be limited to $40mm < w < 100mm$ and $70mm < h < 150mm$. The two switches **model battery** and **model plastic casing**, will generate the phone's battery (modeled as PEC) and a 1mm thick casing with a permittivity of 3, respectively.

27.8.7. BiQuad

The BiQuad Antenna is a rather simple antenna with a comparatively high directivity (about 10 dB). Each of the two folded arms have are a wavelength (λ) long and the total structure is placed at a distance of about $\frac{\lambda}{4}$ over an infinite ground plane. By entering the resonance frequency (default IMS band 2.45 GHz) the length of the two antenna arms are calculated automatically to resonate at λ. The two switches **lumped port excitation** and **coaxial excitation** allow to define the excitation scheme. By default, a lumped port is used to save simulation time, due to the less stringent discretization requirements of this port type. In case of a practical BiQuad antenna a coaxial feeding line should be used.

27.8.8. Patch Antenna Array

The patch array is the most complex antenna template. By defining the patch size, as there is the length l and the width w of the patch and the spacing (hdx and wdy) between the patches, an $n_x \times n_y$ array is set up. The numbers of elements n_x in x-direction and n_y in y-direction are limited due to the computational resources available. An array of 10x10 Elements and the default settings needs less than 5 minutes on a P4 architecture. The beam can be tilted by specifying the two angles in spherical coordinates theta (θ) and phi (φ). Please note that for tilted beams (time delays for the ports) a correct s-parameter evaluation is not possible. If the switch **assign common port for all elements** is turned off, every element will be assigned its own port number, which allows to compute the total coupling using the $n_x \times n_y$ S-matrix.

27.9. Templates - Filters

These templates create Chebycheff filter layouts in microstrip or triplate technology. The Chebycheff order, fractional band width and ripples can be specified, as well as resonance frequency, feed and filter line impedance. The coupling length can be set parametrically for sweeping and optimization purposes.

Hint: Application Note

More information is available in Chapter 29.2.9

27.9.1. Coupled Lines

The layout contains straight quater wavelength coupling sections. The coupling distance is determined from the even and odd impedances which are derived form the coupling coefficients.

27.9.2. Coupled Hairpin

The layout contains coupling sections which are folded in U-shapes in order to save layout space.

27.9.3. Tapped Hairpin

This layout is similar to Coupled Hairpin layout where the first and last coupling section is replaced by a tapped input and output.

27.10. Templates - Thermal

This template defines an environment for a power source with the size of an SMD component. The simulation solver is set to Thermal, only.

27.11. Templates - Environments

This template sets up a number of groups and property definition for multilayer structures.

27.12. Tutorials

By selecting one of the tutorials an empty structure is used and a PDF document is opened which explains step-by-step the necessary actions to build up a simulation task.

Tutorials are stored in the installation folder if **Examples** were selected in the installation process.

27.12.1. Tutorial quick reference guide

1. Start Empire from Desktop
2. Select Tutorials
3. Choose 3D Design - 01 Patch Antenna
4. Click OK Now a PDF is opened which contains instructions to build a model from scratch

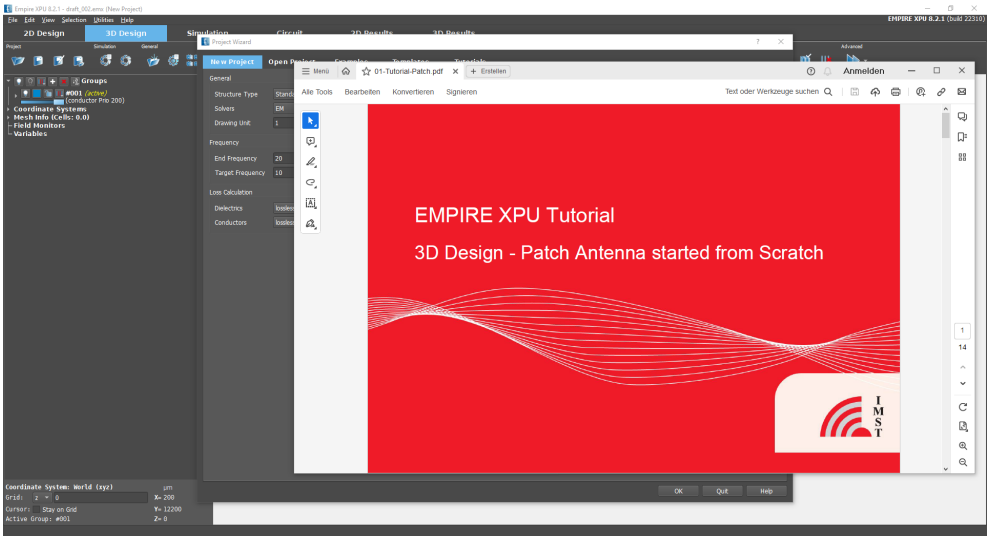


Figure 27.6.: Patch Antenna Tutorial

28. Scripting

EMPIRE XPU™ supports an interface to Python scripts. The **Python** programming language, commands and syntax is available at **www.python.org**. It can be used to:

- Create user-defined elements or material distributions
- Create input, setups and environments
- Run simulations from command line or batch file
- Evaluate results with user-defined postprocessing

Hint: Python scripts

Python scripts (ending **.py**) have to be located in the respective project folder. It is advisable that those folders do not contain spaces or other special characters

28.1. User defined elements

28.1.1. User defined polygons

Python files have to be located in the project folder.

To create a user defined polygon:

1. Create a group with property, e.g. **userpoly anyname $amp**
2. Create a two point polygon with start and end point of the shape

A typical python script, named **userdef.py** to generate a user defined shape has the following content:

```
from math import *                      Include libraries
from numeric import *

def anyname (pts,args):                 function call from userpoly name in EMPIRE XPU™
    p2d = []                            Initializing a 2 dimensional point list
    (x0,y0),(x1,y1) = pts[:2]           Assigning points from a 2-point polygon
    Amp = eval(args[0])                 Every time the drawing is updated this value is renewed

    LX  = int(x1-x0)                    Interval size

    for x in arange(0,LX,LX/100.):      Loop over x values
        y = Amp *(x/LX)**2              function with parameters
        p2d.append((x+x0,y0+y))         Append points to the list

    p2d.append((x1,y0))                 Append last point
    p2d.append((x0,y0))                 Append first point

    return p2d                          Return point list to EMPIRE XPU™
```

28.1.2. User defined library element

- All library objects are available as readable Python files, e.g. `regpoly.py` which can be found in the installation folder under `empire\emplib`
- Copy the Python file into your project folder and rename it to `userelement1.py`
- Rename the class name from, e.g. `class regpoly` to `class user1`
- Replace the name, e.g. `regpoly` by `user1` where appropriate
- Modify the code to add the object to the library

As an example below is a modified Python file (userelement1.py) which has been adapted from the existing element `regpoly.py`. It creates a star instead of a regular polygon.

```
from library import *
from math import *
from ganlib import point_label
class user1_d3d:
    d3d_cylinder_input          = True
    d3d_lbl_def_pts             = [point_label("Left Click on Center Point","P0"),
                                   point_label("Radius of Enclosing Cylinder","db/2"),
                                   point_label("Height (Long Click for Group Height)","z")]
class user1(user1_d3d, element):
  name    = "Star Polygon"
  npoints = 2
  c1 = [3,4,5,6,7,8,10,20]
  c2 = [0.1,0.2,0.25,0.5,0.75]
  parameters  = PARGROUP( store = "LINES",name="Star Parameters",contents = [
        PARGROUP( text  = "Geometry", contents =[
        PARFLOAT( name = "N", text="Number of outer edges N>2", default="5", choice=c1 ),
        PARFLOAT( name = "ri", text="Relative inner radius", default="0.5", choice=c2 ),
         ])])
  def create_objects(s):
      gm    = s.gmf()
      (x0,y0,x1,y1) = s.get_2d_points()
      rad = sqrt((y1-y0)*(y1-y0)+(x1-x0)*(x1-x0))
      off = atan2((y1-y0),(x1-x0))
      ang = 2*pi/s.N
      p2d = []
      p2d.append((x1,y1))
      for i in range(1,int(s.N)+1):
               xi = x0 + s.ri*rad*cos((i-0.5)*ang+off)
            yi = y0 + s.ri*rad*sin((i-0.5)*ang+off)
            x = x0 + rad*cos(i*ang+off)
            y = y0 + rad*sin(i*ang+off)
            p2d.append((xi,yi))
            p2d.append((x,y))
      gm.poly(array(p2d))
      s.objs =  gm.objects()
```

See Figure 28.1 for the output

Depending on the object type the object is created from a

- List of x,y coordinates `gm.poly`
- List of x,y,z coordinates `gm.triangles`

28.2. Material distributions

28.2.1. Material Script

- Script name, python file has to be available in the project folder
- Geometric property for intersecting objects with different material

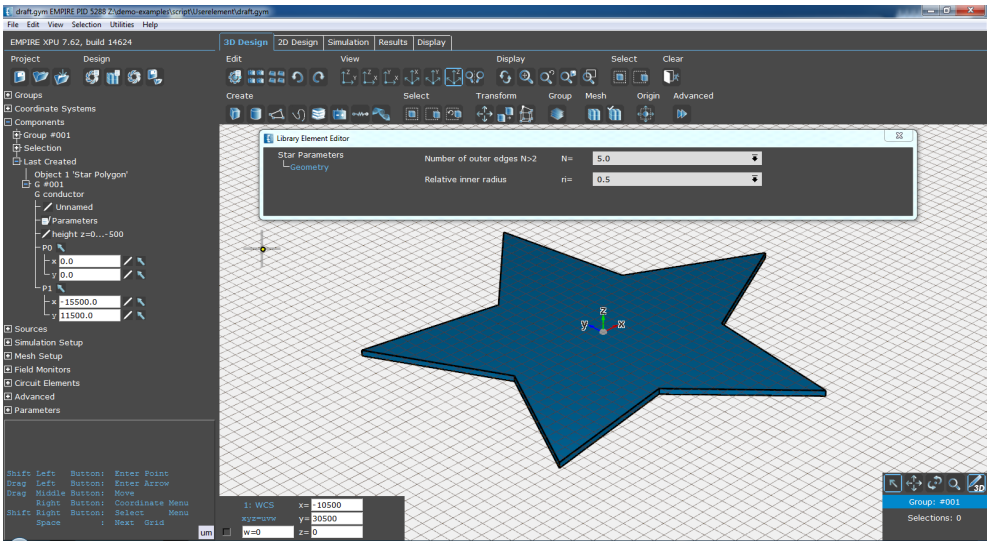


Figure 28.1.: Userdefined Element example

As an example, the script could be defined as follows:

```
def sigma(s):                                          Define conductivity as a function of s

    x = s.X0 + s.X                                     Size of the material box in x-direction
    if x < 0 or s.component != 2: return 0             zero for points outside and x, y components
    else:                         return x / 7000.0    Assigning conductivity for z component

def epsr(s):                                           Define permittivity as a function of s

    return 1.0                                         Set to 1 if not used
```

Meaning of the variables:

- **s.pos0** or **s.X0,s.Y0,s.Z0**: position of Box lower corner (drawing units)
- **s.pos** or **s.X,s.Y,s.Z**: position inside Box (drawing units)
- **s.index0** or **s.x0,s.y0,s.z0**: index of Box lower corner
- **s.index** or **s.x,s.y,s.z**: index inside Box
- **s.component**:0,1,2 field component x,y,z

Plasma Script

- Script name, python file has to be available in the project folder
- Geometric property for intersecting objects with different material

28.3. Scripted input, setups and environments

There are 2 interfaces available for scripted input:

- **KMF** - Interactive object handling, extended capabilities (Kallisto Meta File). It supports script based functionalities, e.g.:
  - Copy, mirror and rotate objects
  - Selection, tagging and referencing concept for created objects
  - World Coordinate System (WCS) shift
  - Import of layout, faceted and BREP CAD data, including options

  An on-line help is available in **Help** - **Scripting** for the KMF scripting interface
- **GMF** - One-directional object handling, functional capabilities (Ganymede Meta File). This interface supports script based functionalities for creation of structures, e.g.:
  - Creation of objects (box, polygon, surface, STL-import)
  - Layer/group setup

  The following sections describe the GMF scripting capapilities and a function list in section 28.6.2.

If the graphical user interface is opened a script can be executed from the Menu item **File** - **Run Script**. This can be useful for automated input, setups or environments. Examples are given in the below subsections. (The content of the boxes should be saved as a file called `script.py` and located in the project folder).

The list of supported functions is available in Chapter 28.6.

28.3.1. Set Simulation Parameters

The following example sets the boundary condition at zmin to electric, adds additional space at zmax with 5000 units, sets the maximum frequency to 15 GHz, energy criterion to 70 dB and an equidistant meshing with 100 units.

```
from scripting import *
g = gmf()                              # Initialize
set_fs_parameter("zmin","electric")    # Boundary condition
set_fs_parameter("zmax_as","5000")     # Additional space
set_fs_parameter("F_END",15e9)         # Highest Frequency
set_fs_parameter("chksum",70)          # Energy decay
set_auto_parameter("disx",100)         # xmesh equidistant
g.load()                               # Execute
```

All available parameters which can be set with a script can be found in each project file ending `.par`.

In order to find relevant parameter and value:

- Load a new (empty) project and save as draft.emx
- Modify the settings (e.g. set Structure Type to Medium Q) and save as modify.emx
- Compare the files draft.par and modify.par

In this example it can be found that the modified parameter is named **TYPE** and the value has changed from **SHR** to **MED**.

28.3.2. Create Layer Stack

The following example defines several groups with predefined heights and electric properties.

```
from scripting import *
g = gmf()                              # Initialize
g.layer("metal","red")                 # Group name, color
g.layerextrude("z",800,810)            # Group height
g.layertext("conductor")               # Group property
g.layer("substrate","green")           # Group name, color
g.layerextrude("z",0,800)              # Group height
g.layertext("dielectric epsr 2.2")     # Group property
g.load()                               # Execute
```

Note that the last defined layer will be the active group.

28.3.3. Create objects

The following example defines several objects using the group height.

```
from scripting import *
g = gmf()                          # Initialize
g.layer("metal","red")             # Group name, color
g.layerextrude("z",800,810)        # Group height
g.layertext("conductor")           # Group property
g.box(-4500,-6000,4500,6000)       # Box with x0,y0,x1,y1
g.circle(0,0,2000)                 # Cylinder x,y,r
g.poly(0,0,0,3000,2000,0)          # Polygon with x0,y0,x1,y1,x2,y2,...
g.load()                           # Execute
```

It is also possible to define loops, parameterize values or apply operations.

28.3.4. Create and modify objects

The following example defines several operations with objects using the push command (holes inside a polygon)

```
from scripting import *
g = gmf()                           # Initialize
g.layer("metal","red")              # Group name, color
g.layerextrude("z",800,810)         # Group height
g.layertext("conductor")            # Group property
g.box(-4500,-6000,4500,6000)        # Box with x0,y0,x1,y1
g.push("box1")                      # Select 1st box
g.box(0,0,8000,8000)                #
g.push("box2")                      # Select 2nd box
g.merge("box1 box2")                # Union operation
g.push("xx")                        # Select result
circles = ""                        # Initialize string
for i in range(0,5):                # Loop
    g.circle(i*500,i*500,200)       # Create cylinder
    g.push(str(i))                  # Select cylinder
    circles = circles+str(i)+" "    # Extend string
g.subtract("xx %s"%circles)         # Subtract
g.load()                            # Execute
```

Note that each **push** operation requires a unique text string.

28.3.5. Create ports and monitors

The following example defines a complete simulation setup ready for simulation.

```
from scripting import *
g = gmf()                                           # Initialize
set_fs_parameter("zmin","electric")                 # Boundary condition
set_fs_parameter("TYPE","ANT")                      # Structure Type
g.layer("metal","red")                              # Group name, color
g.layerextrude("z",800,810)                         # Group height
g.layertext("conductor")                            # Group property
g.box(-4500,-6000,4500,6000)                        # Box with x0,y0,x1,y1
g.layer("substrate","green")                        # Group name, color
g.layerextrude("z",0,800)                           # Group height
g.layertext("dielectric epsr 2.2")                  # Group property
g.box(-10000,-10000,10000,10000)                    # Box with x0,y0,x1,y1
g.layer("port","blue")                              # Group name, color
g.layerextrude("z",0,800)                           # Group height
g.pp_port(-1600,-200,-1200,200,num=1,do_excite=1)   # Perpendicular lumped port x0,y0,x1,y1
g.farfield(0,0,1000,1e10)                           # Far field monitor at x,y,z and 10 GHz
g.load()                                            # Execute
```

More examples can be found in the templates scripts (e.g. msl_line.py) located at
`C:\EMPIRE_XPU_9.x.x\empire\ganymed_2\templates`

28.4. Scripted Simulations

EMPIRE XPU™ can be started from the command line or using a batch file (.bat) in Windows or a shell script in Linux.

In the following the installation folder `EMPIRE_XPU_9.x.x\empire` is abbreviated by `XPU` and a working folder contains

- an input file **draft.emx** (at least an empty input file draft.emx has to be available, created e.g. with **New Project**).
- a project folder **draft** including a script file **script.py**
- a batch file **run.bat** or shell script **run.sh**

Example for a run.bat file (Windows):

```
"C:\XPU\PY3k-W64\Python3k\python.exe" "-O" "-u" "C:\XPU\env.py" script draft.emx script.py
```

Example for a run.sh file (Linux):

```
$HOME/XPU/PY3k-64/bin/python -O -u $HOME/XPU/env.py script draft.emx script.py
```

The script for running a simulation on the local machine would look like this:

```
from scripting import *
g = gmf()                  # Initialize
                           # e.g. set parameters
g.load()                   # Execute
simulate()                 # Start Simulation
```

It is also possible to create a new folder and run the simulation there:

```
from scripting import *
g = gmf()                  # Initialize
                           # e.g. set parameters
g.load()                   # Execute
save(sim/patch.gym)        # Save and switch to folder
simulate()                 # Start Simulation
```

If an Empire Server is running on a remote machine (e.g. hostname=imst4060 or IP address) the simulation can also be carried out on the host:

```
from scripting import *
g = gmf()                  # Initialize
                           # e.g. set parameters
g.load()                   # Execute
batch(hosts="imst4060")    # Start Simulation
```

28.5. Scripted Postprocessing

If a script called **postproc.py** is available in the project folder it will be executed automatically during the postprocessing. Other scripts can be run manually with the aid of the Menu **File - Run Script** for postprocessing.

As a simple example the following script reads the S-parameters `s11` and `s21` (frequency, magnitude, phase) and creates `s11_reim` and `s21_reim` files (frequency, real, imaginary)

```
from Numeric import *
import os
def reim(file):
     dat      = DATA(file)
     fp       = open(file+"_reim","w")
     for i in range(len(dat.index)):
        freq  =  dat.index[i]
        real  =  dat.data[i].real
        imag  =  dat.data[i].imag
        print >>fp,"%.15e %.15e %.15e"%(freq,real,imag)
     fp.close()
if subdir == "sub-1":
     reim(os.path.join(path,"s1_1"))
     reim(os.path.join(path,"s2_1"))
```

The following postprocessing script **postproc.py** automatically removes the field recording file which is used for far field transformation and can be very large. (Note, after removing the file successive far field transformations are no longer possible)

```
import glob,string,os

mask = 'Farfield_1'                    File to be deleted
cwd = os.getcwd()                      get current working folder
os.chdir(path)                         change to simulation folder
for fn in glob.glob(mask):
    dprint('Delete: {}'.format(fn))    print to log window
    os.remove(fn)

os.chdir(cwd)                          reset current working folder
```

28.6. Supported Functions

The EMPIRE XPU™ scripting modules (scripting.py, gmf.py) can be used in scripts with
`from scripting import *`

Scripting includes geometry generation and manipulation, simulation, batch simulation and result based calculation. There are two ways of starting a script:

- From the File/Run Script menu. This is the reommended way for script-development
- From the command line[1] or from another tool with the commands listed in 28.4

[1] Start cmd.exe from Windows. Copy the command from the empire-startup icon and adjust the command

28.6.1. Functions in scripting.py

- set_auto_parameter(name,val):
  set autodisc parameter
  some interesting parameters are
  disx,disy,disz:
  Mesh setting, e.g. auto, value (in units)
  disx_opt,disy_opt,disz_opt:
  Mesh hint, e.g. off, min 10, max 100 (in units)

- set_fs_parameter(name,val):
  set simulator parameter
  some interesting parameters are
  xmin,xmax,ymin,ymax,zmin,zmax:
  Boundary Conditions, e.g. electric, magnetic, pml 6, sheet 1
  dx:
  Drawing unit in m
  F_START:
  Start Frequency in Hz
  F_END:
  End Frequency in Hz
  THICKNESS:
  Flat metal thickness in cells (autodisc)
  PGA:
  0: off, 1: 2d, 2: 3d, 3: 2d+3d
  MATERIAL:
  LL: lossless, D1: one pole lossy, D2: two poles lossy
  METAL:
  LL: lossless, D1: one pole lossy, D2: two poles lossy
  autoreso:
  Autodisc resolution, values:
  0,5,10,15,20,25,30 ....

- set_parameter(name,val):
  set empire parameter (user defined),
  example:
  set_parameter("a",1000)

- set_parameters(**kw):
  set empire parameters (user defined),
  example:
  set_parameters(a=1000,b=200,e=400)

- copy_file(src,dst):
  file copy utility
  example:
  copy_file("from.gym","to.gym")

- import_gdsii(fn, scale=1):
  import gdsii file
  example:

```
import_gdsii("test.gdsii", scale=1e3)
```

- load(fn,is_auto=1):
  load empire structure and change to its directory
  example:
  load("here/test.gym")

- convert(fn):
  autodisc (if not set to manual)
  save and convert structure and change to its directory
  example:
  convert("here/test.gym")

- save(fn):
  save structure and change to its directory
  example:
  save("here/test.gym")

- simulate(do_save=1, do_automesh=1):
  (autodisc, save and convert structure and change to its directory)
  and perform simulation.
  With do_save=1 the same as pressing the "simulate" button.
  example:
  simulate()
  or:
  convert("here/test.gym")
  simulate(do_save=0)

- postproc():
  Perform a postprocessing

- batch(do_automesh=True,batch_no=1,hosts=None,wait=True):
  Start batch processing, batch has to be properly set up,
  e.g. manually or with the setup_batch function.
  Does File Generation before Simulation
  example:
  load("test.gym")
  batch()

  batch_no: 1..... or None
  None: append Batch to Batch List, max 128 Batches

  hosts: "hostname.xx.yy" or "ip-addr" or "hostname" or list of this

  Wait: wait for finish OR return function to call and check for finish.
  Call this function regulary !!!

- autodisc():
  Perform autodisc, e.g. if converting a structure where autodisc is
  set to manual.

```
example:
    load("test.gym")
    autodisc()
    simulate()
```

- gmf(*args,**kw):
```
generate a metafile Object (see doc for gmf.py)
, load it into structure with the load method.
example:
     g = gmf()
     ........    Generate Geometry Objects
     g.load()
```

- empty_disc(dirs=(0,1,2)):
```
Erase all discretisation lines
example:
     empty_disc()        #    All directions
     empty_disc([0])     #    only x direction
     empty_disc([1,2])   #    only y and z direction
```

- get_polygons(layers=None):
```
Get a list of all polygons on certain layers,
each polygon contains a list of 2-tuples which are the polygon points.
layers is a list of layer names.

example:
     lp = get_polygons(["GND"])
     lp = get_polygons(["userdef1","userdef2"])
```

- get_polygons_bbox(layers=None):
```
get bounding box of all polygons,
see get_polygons.

The Bounding box is a [x0,y0,x1,y1] list,
if all polygons are in the xy plane.
```

- setup_ypar(name,nports):
```
Set up a Y and S parameter postprocessing.
name:   Filename for .snp file
nports: Number of ports
```

- setup_batch(ports,hosts):
```
 Set up a batch processing
 example (for 10 ports):
     setup_batch(10,"host1 host2 host3")
```

- setup_postproc(*args):
```
Setup Postprocessing
```

- db20(x):
  Convenience function for dB calculations.

- find_maximum(filename,min=-1e100,max=1e100):
  Find the maximum absolute value in a data file,
  restricted from min to max.

  example:
  frequency = find_maximum("sub-1/s1_1",min=1e9,max=10e9)

- find_minium(filename,min=-1e100,max=1e100):
  Find the minimum absolute value in a data file,
  restricted from min to max.

  example:
  frequency = find_maximum("sub-1/s1_1",min=1e9,max=10e9)

- results(filename,position=None):
  Extract a data point from a data file with interpolation
  example:
  frequency = find_maximum("sub-1/s1_1")
  peak_height = db20(result("sub-1/s1_1"))

- efficiency(file="sub-1/nf2ff.log"):
  Extract the antenna efficiency from a log file

28.6.2. Functions in gmf.py

- points(s,*args):
  store 2d points for later use of poly, rotpoly etc.
  example:

```
g = gmf()
g.layer("C01","red")     #   use extrusion from (existing) layer
g.points(x0,y0,x1,y1)
g.poly()
g.load()
```

- points3d(s,*args):
  store 3d points

- triangles(s,*args):
  store 3d triangles for later use of makesolid
  example:

```
g = gmf()
g.layer("C01","red")     #   use extrusion from (existing) layer
g.triangles(x0  ,y0,z0,x1  ,y1,z1,x2  ,y2,z2);    #   store first triangle
g.triangles(x0+1,y0,z0,x1+1,y1,z1,x2+1,y2,z2);    #   second  triangle
g.solid()
```

```
g.load()
```

- solid(s,*args,**kw):
  generate a solid (a triangle confined 3d object or a 2d triangle tesselated sheet)

```
example:
    g = gmf()
    g.layer("C01","red")     #   use extrusion from (existing) layer
    g.triangles(x0  ,y0,z0,x1  ,y1,z1,x2  ,y2,z2);    #   store first triangle
    g.triangles(x0+1,y0,z0,x1+1,y1,z1,x2+1,y2,z2);    #   second  triangle
    g.solid()
    g.load()
```

- radius(s,*args):
  internal use only

- box(s,x0,y0,x1,y1):
  generate a box from 4 2d coordinates and from the current extrusion

```
example:
    g = gmf()
    g.layer("C01","red")
    g.layerextrude("z",800,810)    #     set layer's extrusion to z=800..810
    g.layertext("conductor")       #     set layer's property
    .......

    g.layer("C01")
    g.box(0,0,100,100)
    g.load()
```

- poly(s,*args):
  generate an extruded polygon 2d coordinates and from the current extrusion

```
example:
    g = gmf()
    g.layer("C01","red")
    g.layerextrude("z",800,810)    #     set layer's extrusion to z=800..810
    g.layertext("conductor")       #     set layer's property
    .......

    g.layer("C01")
    g.poly(0,0,20,-30,100,100)     #     A triangle,  autmatically closed
    g.load()
```

- circle(s,x0,y0,r):
  generate an extruded circle

```
example:
    g = gmf()
    g.layer("C01","red")
    g.layerextrude("z",800,810)    #     set layer's extrusion to z=800..810
    g.layertext("conductor")       #     set layer's property
    .......
```

```
g.layer("C01")
g.circle(0,0,100)
g.load()
```

- rotpoly(s,x0,y0,z0,x1,y1,z1):
  generate a rotated polygon (points done with points)
  example:

```
g = gmf()
g.layer("C01","red")
g.layerextrude("z",800,810)     #     set layer's extrusion to z=800..810
g.layertext("conductor")        #     set layer's property
.......
g.layer("C01")
g.points(0,0,-20,50,100,100)
g.rotpoly(0,0,0,100,0,0)        #     Rot-Axis must be 1 dimensional vector

g.load()                        #     in the extrusion-plane
```

- strip(s, width, mode, *args):
  generate a strip with a given width along a set of points
  example:

```
g = gmf()
g.layer("C01","red")
g.layerextrude("z",0,10)    #     set layer's extrusion to z=0..10
.......
g.strip(10, "Strip",                        0,0,     20,-30,     100,100)
                                    #   create strip of width 10
g.strip(10, "Closed Strip",             100+0,0, 100+20,-30, 100+100,100)
                                    #   create closed strip of width 10
g.strip(10, "Circular Strip",           200+0,0, 200+20,-30, 200+100,100)
                                    #   create strip of width 10 with rounded edge
s
g.strip(10, "Circular Closed Strip", 300+0,0, 300+20,-30, 300+100,100)
                                    #   create closed strip of width 10 with round
ed edges
g.load()
```

- parameter(s,name,min=-1000,max=1000,step=200,val=0,opt=1,comment=''):
  generate a parameter, must be done (with g.load() !) before usage of term()
  opt: optimisation flag
  example:

```
g = gmf()
g.parameter("a",min=0,max=1000,step=100,val=100,opt=0)
g.load()
```

- equation(s,name,val="3.1415"):
  generate a equation, must be done (with g.load() !) before usage of term()
  opt: optimisation flag
  example:

```
g = gmf()
g.equation("a",val="3*b")
```

```
g.load()
```

- layer(s,name,color="",visibility=1,opaque=None,autodisc=""):
  generate a layer or switch to a layer
  name: layer name
  color: color e.g. "red","#25f","#239089" (rgb, hex values)
  visibility: 0 or 1
  opaque: 0 to 1.0(fully transparent)
  autodisc: 0 important, 1 not important, 2 ignore, 3 dense

- addmesh(s,dir,*args):
  add fixed lines to the current mesh in direction
  example:
  g = gmf()
  g.addmesh("x",0, 1000, 2000)
  g.load()

- deletemesh(s,dir):
  delete mesh lines in direction
  dir: "x" "y" or "z"

- unselect(s):
  unselect everything, if objects were selected before

- layerextrude(s,dir,c0,c1):
  set layer extrusion
  dir: "x" "y" or "z"
  c0: min coordinate
  c1: max coordinate

- objectextrude(s,dir,c0,c1):
  set object extrusion
  dir: "x" "y" or "z"
  c0: min coordinate
  c1: max coordinate

- layertext(s,text):
  set layer's property

- layerselect(s,name):
  select all objects on layer

- layersubstract(s,name):
  substract all layer's objects from selection

- layerset(s,name, keep=True, assgn=True):
  move or copy selected objects to given layer

- objecttext(s,text):
  set object's property

- objecttexts(s,*texts):
  set object's property(s)

- push(s,text):
  select last created object

- viaselect(s, rmin=None, rmax=None):
  select all vias of given radius range in current layer

- merge(s,text=""):
  merge selected objects (see push)

- subtract(s,text):
  subtract selected objects (see push): 1st - rest

- delete(s):
  delete selected objects

- oversize(s, osz):
  oversize selected objects according to given number

- msl_port(s,x0,y0,x1,y1,w,hb,num,epsr=1,t=1,is_conc=1,do_excite=1,ht=None,R=50,Rcopy="Off",pd=None,slot=None,mode=None):
  generate msl port with current extrusion
  x0,y0: start coordinate
  x1,y1: end coordinate (only x0!=x1 or y0!=y1 not both)
  w: width
  t: conductor thickness
  hb: bottom height
  epsr: permittivity (only for ZL calc !!!)
  is_conc: use concentrated port ?
  do_excite: excitation switch
  R: Load impedance
  Rcopy: Use for Postprocessing
  pd: Measurement Distance
  mode: None,"odd","even" Coupled Lines
  slot: coupled line slot width

- tpl_port(s,x0,y0,x1,y1,w,hb,ht,num,epsr=1,t=1,is_conc=1,do_excite=1,dly=0, WV=1.0, WC=1.0,WE=1,R=50, Rcopy="Off",slot=None,mode=None):
  generate stripline port with current extrusion
  x0,y0: start coordinate
  x1,y1: end coordinate (only x0!=x1 or y0!=y1 not both)
  w: width
  t: conductor thickness
  hb: bottom height

```
            ht:         bottom height
            epsr:       permittivity  (only for ZL calc !!!)
            is_conc:    use concentrated port ?
            do_excite:  excitation switch
            mode:       None,"odd","even"               Coupled Lines
            slot:       coupled line slot width
```

- cpw_port(s,x0,y0,x1,y1,w,slot,b,num,epsr=1,t=1,is_conc=1,do_excite=1):

```
        generate coplanar port with current extrusion
            x0,y0:      start coordinate
            x1,y1:      end   coordinate  (only x0!=x1 or y0!=y1 not both)
            w:          width
            t:          conductor thickness
            slot:       slot width
            b:          complete width
            epsr:       permittivity  (only for ZL calc !!!)
            is_conc:    use concentrated port ?
            do_excite:  excitation switch
```

- lumped_port(s,x0,y0,x1,y1,w,num=1,do_excite=1,dly=0,WV=1,WC=1,WE=1,R=50):

```
        generate lumped port with current extrusion
            x0,y0:      start coordinate
            x1,y1:      end   coordinate  (only x0!=x1 or y0!=y1 not both)
            w:          width
            num:        port number
            is_conc:    use concentrated port ?
            do_excite:  excitation switch
            WV:         voltage    weight
            WC:         current    weight
            WE:         excitation weight
            R:          load
            dly:        time delay
```

- wire_port(s,x0,y0,z0,x1,y1,z1, d=0.015,num=1,do_excite=1,dly=0,WV=1,WC=1,WE=1,R=50):

```
        generate wire port
            x0,y0,z0:   start coordinate
            x1,y1,z1:   end   coordinate  (only x0!=x1 or y0!=y1 not both)
            d:          wire diameter
            num:        port number
            do_excite:  excitation switch
            WV:         voltage    weight
            WC:         current    weight
            WE:         excitation weight
            R:          load
            dly:        time delay
```

- pp_port(s,x0,y0,x1,y1,w=0,num=1,do_excite=1,dly=0,WV=1,WC=1,WE=1,R=50,fref_dly=None):

```
        generate perpendicular port with current extrusion
            x0,y0:      start coordinate
            x1,y1:      end   coordinate  (only x0!=x1 or y0!=y1 not both)
```

```
num:       port number
is_conc:   use concentrated port ?
do_excite: excitation switch
WV:        voltage    weight
WC:        current    weight
WE:        excitation weight
R:         load
dly:       time delay
fref_fly:  time delay ref freq. or None
```

- coax_port(s,x0,y0,x1,y1,di,dd,da,num,epsr,pra=80,prd=100,pri=120,is_conc=1,do_exc ite=1):

```
generate coaxial port with current extrusion
   x0,y0:     start coordinate
   x1,y1:     end   coordinate  (only x0!=x1 or y0!=y1 not both)
   num:       port number
   di:        inner diameter
   da:        outer diameter
   dd:        dielectric diameter
   epsr:      permittivity
   is_conc:   use concentrated port ?
   do_excite: excitation switch
```

- wg_port(s,x0,y0,x1,y1,w,h,dw,num,is_conc=1,do_excite=1,**kw):

```
generate waveguide port with current extrusion
   x0,y0:     start coordinate
   x1,y1:     end   coordinate  (only x0!=x1 or y0!=y1 not both)
   num:       port number
   w:         width
   h:         height
   dw:        wall thickness
   is_conc:   use concentrated port ?
   do_excite: excitation switch
```

- cwg_port(s,x0,y0,x1,y1,d,dw,num,amd=0,phi=0,nsk=1,is_conc=1,do_excite=1,exc=None, **kw):

```
generate circular waveguide port with current extrusion
   x0,y0:     start coordinate
   x1,y1:     end   coordinate  (only x0!=x1 or y0!=y1 not both)
   num:       port number, e.g 1 , '1-2'
   d:         Diameter
   dw:        wall thickness
   is_conc:   use concentrated port ?
   do_excite: excitation switch
   exc:       Excite term, e.g.  1 , '1-2'
   amd:       use Analytic Modes  (default 0 or 1)
   phi:       tilt Angle for Analytic Modes       (default:  0      )
   nsk:       first mode Number for Analytic Mode (default:  0 TE11 )
```

- qtem_port(s,x0,y0,z0,x1,y1,z1,num,do_excite=1,**kw):

```
generate qtem port with current extrusion
   x0,y0,z0:     ref point
```

```
            x1,y1,z1:     point away in prop direction
            num:          port number, e.g 1
            do_excite:    excitation switch
```

- field_volume(s,dir_pos,freqs,fk="Exyz",scale="log_70",loop="phase_20_deg",border="black",btype="equi_15",norm="ef",visible=1,what="EMFullVolume"):
  Create EM field volume monitor

- field_point(s,x,y,z,f0=None,f1=None,nf=None,options="enable,eabs,habs,ex,ey,ez,hx,hy,hz,itrans,ilong",what="EMPoint",**kw):
  Create EM field point monitor

- thermal_monitor(s,dir_pos,mode="maximum",keys=15,unit="K",min="auto",max="auto",equi=15):
  Create Thermal monitor

- farfield(s,x,y,z,freqs,visible=1,options=()):
  Create EM far field monitor

- load(s,to_stack=0,with_undo=0,return_objs=0):
  load objects into the structure

- save(s,fn):
  save metafile

- import_stl(s,fn):
  Import STL file

- wire(s,d,n=20,dynamic=1):
  generate a circular wire object to draw wires like bonds etc.
  d: diameter, n: resolution
  example:

```
g = gmf()
w = g.wire(100)                 #     make a wire object
w.add((x0,y0,z0),(x1,y1,z1))    #     add 2 points
w.write()                       #     write wire to gmf
```

- vector_text(s,txt,x0,y0,x1,y1,hd=1.1,size=None,font="default",return_polys=False,arrow=None,layer=None,**kw):
  generate vector text

```
txt:     the text string
x0,y0:   start coordinate
x1,y1:   end   coordinate
hd:      head  distance factor (vertical)
size:    vertical size (if given)
```

Part III.

Application Guide

This chapter describes the work flow for typical and advanced applications.

- In **EM (FDTD) Simulation** some typical applications are listed in alphabetical order and recipes are given on how to obtain the desired results.
- More **Advanced Applications** and their usage can be found in the following chapters
  - **Circuit Simulation** describes the setup, usage and background of circuit simulations
  - **Thermal Simulation** describes the setup, usage and background of thermal simulations
  - **Body Model Simulation** explains the usage of voxel files including the voxel editor and posing module

29. EM Simulation

The flow for a basic simulation can be described by the following steps

1. Start EMPIRE XPU™
   - Double click desktop icon and select **New Project**, optionally enter simulation parameters and start from scratch.
   - Create a new project folder, using **File - Save As**
2. Enter components in Groups
   - Create or modify objects, chapters 15 and 16
     - Use current or create a new group
     - Use the **Create toolbar** buttons for new objects
     - Optionally, select objects and apply operation buttons
   - Import data, sections 17.1.4 and 17.1.5
     - Import 2D layout data files and assign group height
     - Import 3D solid data files
3. Set properties of Groups, chapter 18
   - For EM simulations define electric properties
   - For Thermal simulations define thermal properties
   - Optionally, adjust color and display style
4. Enter sources, chapter 20
   - For S-Parameter calculations use ports
   - For thermal simulations use thermal sources
   - Optionally, use a Plane Wave source as excitation
5. Set simulation parameters, chapter 23
   - Click button  and check settings
6. Optionally, define Field Monitors, chapter 19
7. Start the simulation , chapter 24

29.1. Basic Applications

The following section describe specific settings for different applications.

29.1.1. ACD calculation (Averaged Current Density)

Usage

- Define excitation scene (antenna, plane wave, ...)
- Create objects with conductivities or use a voxel file, see Chapter 32
- Create an EM Field Monitor, Section 29.1.16, covering a volume large enough for averaging, minimum one cell thickness
- Adjust display parameters, Section 29.1.14, and specify settings (target frequency, interpolation)
- Run the simulation
- Switch to 3D Display mode

Remarks

- Check settings in the animation definition (e.g. component ACD_1 for $1cm^2$ averaging). The number can be edited.

Examples

- Load **Examples - SAR & ACD - Disc in homogeneous Magnetic field**

29.1.2. Active impedance calculation

Ports with different port numbers can be excited at once. It delivers the diagonal of the S-Matrix, as well as voltages and currents of all ports.

Usage

- Define all ports with different port numbers and excitation On
- In Simulation Setup select "Simulation Mode" to "Simultaneous Excitation (Active Impedance)"
- Run the simulation

Remarks

- After simulation the results for all ports will be available in one subfolder, e.g. sub-1

29.1.3. Capacitance calculation

Usage

- Create a lumped port (number 1), Section 29.1.22, between a capacitive structure, e.g. 2 plates
- Select **Simulation** tab - **Simulation Config - Equations**
- Right click on sub-1 select **Add Equation - Symbolic Equation**, select equation **ZC** from the **Template**
- Start Simulation
- **2D Results**, Plot Type **Impedance** after postprocessing

Remarks

- The capacitance is derived from the input impedance C=Im(Z1.in)*(2pi f)
- Valid for low frequency (linear range of the 1/Im(Z1.in))

29.1.4. Circular Polarization (Antennas, Waveguides)

Narrow band

Usage

- Create an exciting port (number 1), Section 29.1.22, for one polarization
- Create another exciting port (number 2), for opposite polarization and specify a delay which generates a 90 degree phase shift at the target frequency, e.g. `90@10e9`
- Set **Simulation Mode = Simultaneous Simulation** in **Simulation Setup**
- Optionally, select E_{LHCP}, E_{RHCP} or Axial Ratio in the Far field Monitor
- Start Simulation 

Remarks

- Due to the simultaneous excitation only diagonal elements of the S-Matrix can be calculated (Active impedance calculation)
- The phase shift is converted to a time shift of the pulse and therefore only valid for a narrow frequency range

Broad band

Usage

- Create an exciting port (number 1), Section 29.1.22, for one polarization
- Create another exciting port (number 2), for opposite polarization
- Edit PortParameters and set **Broadband Phase Shift** to 90 degrees
- In **Simulation Setup EM Setup** set **AC only simulation** = "on"
- Optionally, select E_{LHCP}, E_{RHCP} or Axial Ratio in the Far field Setup
- Start Simulation 

Remarks

- Due to the simultaneous excitation only diagonal elements of the S-Matrix can be calculated (Active impedance calculation)
- For broadband simulation the phase shift of the second port changes the pulse shape and only valid for modulated pulses (AC only)

29.1.5. Efficiency calculation (Antennas)

Usage

- Create a Far field recording, Section 29.1.8
- Start simulation 

Remarks

- In the log window the efficiency is displayed for each far field transformation
- For each far field calculation the result will also be stored in **.log** files, e.g. to be used in optimization goals
- **Radiation efficiency**: Radiated power over input (accepted) power at input port $\eta = P_{rad}/P_{in}$
- **Total efficiency**: Radiated power over incident (available) power at input port $\eta = P_{rad}/P_{inc}$

The power values are normalized depending on the entered far field normalization in the far field processing setups

- **Directivity Normalisation**: Radiated power $P_{rad} = 1.0$, other power values scale
- **Gain Normalisation**: Input (accepted) power $P_{in} = 1.0$, other power values scale
- **Gain real Normalisation**: Incident (available) power $P_{inc} = 1.0$, other power values scale
- **Excitation Normalisation**: Values in Watt

Hint

For the efficiency the power at the port is evaluated. Strong near fields can induce currents in the port load and effect on the power evaluation. Therefore, in case of unshielded ports the ports should be located away from the reactive near field or should be as small as possible (lumped ports)

29.1.6. Far field animation

Usage

- Create far field monitor before simulation, Section 29.1.8
- Optionally, adjust settings (e.g. Sweep mode: 3D Sweep) in the **Processing Setups**, Section 19.3.2
- Adjust **Plot Options** (result file name, style, size, position, ...)
- Exit with OK, Switch to **3D Results** Tab

Remarks

- The file name convention contains frequency, and component
- The file contains Sweep Angles Theta, Phi, Magnitude, Phase
- A prefix may be defined in the file name to distinguish further results

Examples

- Load **Tutorials - Patch Antenna**

29.1.7. Far field plot

Usage

- Create Far Field Monitor before simulation, Section 29.1.8
- If not already defined, adjust settings (e.g. Sweep mode: 2D Fixed Phi) in the in the **Processing Setups**, Section 19.3.2
- After simulation and postprocessing select **2D Results** Tab, select **Plot Type** = **Far fields**

Remarks

- The file name contains Frequency, Cut Angle, and Component
- The file contains Sweep Angle, Magnitude, Phase
- A prefix may be defined in the file name to distinguish further results

Examples

- Load **Tutorials - Patch Antenna**

29.1.8. Far field recording

Usage

- Create a Field Monitor **EM Far field**  for far field definition
- In **Storage Options**, enter frequency points
- Before or after simulation adjust settings in the **Processing Setups** sections
- For results see Far field plot, Section 29.1.7, or Far field animation, Section 29.1.6

Remarks

- The result file `Farfield_1` (default) is created after simulation which contains the nearfield on a box surface (binary file)
- The far field transformation is carried out in post processing

Examples

- Load **Tutorials - 01 Patch Antenna**

29.1.9. Gain calculation (Antennas)

Usage

- Create a Far field recording, Section 29.1.8
- In Field Monitor - EM Farfield - Processing Setups - Normalization can be set
- Account for losses in the structure, only:
  - Use **Type - Gain**
- Account for reflected power at reference port:
  - Use **Type - Gain_Abs**
- Start postprocessing

Remarks

- Gain calculation make use of the incident power at ports
- The program can determine the exciting ports automatically **Gain Port automatic**
- For special purposes the reference port can also be defined by the user

29.1.10. Impedance calculation

Usage

- Create Port(s), Section 29.1.22
- Enter unique port numbers for multiple ports
- Switch on/off port excitation (minimum one port excited)
- Start Simulation 
- **2D Results** using **Plot Type: Impedance**

Remarks

- Each excited port with unique number n creates a subfolder sub-n
- In folder sub-n the results Zn.in and Zk are available after postprocessing

Examples

- Load **Tutorials - Multi Layer Spiral Inductor**

29.1.11. Inductance calculation

Usage

- Create a lumped port (number 1), Section 29.1.22, between inductive structure, e.g. a coil
- In **Simulation Setup** set **Structure Type** to **Inductor**
- Select **Simulation** tab - **Simulation Config** - **Equations**
- Right click on sub-1 select **Add Equation** - **Symbolic Equation**, select equation **ZL** from the **Template**
- Start Simulation
- **2D Results**, PLot Type **Impedance** after postprocessing

Remarks

- The inductance is derived from the input impedance L=Im(Z1.in)/(2pi f)
- Valid for low frequency (linear range of the Im(Z1.in))

Examples

- Load **Tutorials** - **Multi Layer Spiral Inductor**

29.1.12. Low frequency algorithm

In order to calculate ACD/SAR/EIAV for low frequencies, e.g. 50 Hz, the low frequency algorithm can be applied.

Usage

- Enter the **Gabriel Target Frequency** in the **voxbox** property definition
- Define a dump box for a high frequency, e.g. 10 MHz
- Specify the high frequency as End frequency in the simulation setup
- After simulation specify the low target frequency in the **Plot Options** definition of the field monitor

Remarks

- The Gabriel parameters will be calculated and used in the simulation for the specified low target frequency
- It can be advantageous to set the permittivity to 1 in the voxbox property definition

29.1.13. Multiple port excitation (independent)

Usage

- Create Port(s), Section 29.1.22
- Enter unique port numbers for all ports
- Switch on port excitations
- Subsequent simulation of all ports on the localhost:
  - Click **Start Simulation** 
- Distributed simulations of all ports on remote hosts:
  - **Simulation** Tab - **Host Setup**, Select Hosts
  - Select subfolders
  - Click Start Simulation

Remarks

- The independent port simulation yields the complete scattering matrix

Examples

- Load **Examples - Coupler & Filters - Branch Line Coupler**

29.1.14. Near field animation

Usage

- Create near field monitor before simulation, Section 29.1.16
- Adjust settings in Field Monitor - Plot Options (field, component, hill amplitude, equilines, vector, ...)
- After simulation switch to **3D** Tab

Remarks

- Plot option definitions may be entered after the simulation

Examples

- Load **Tutorials - Low Pass Microstrip Filter**

29.1.15. Near field plot

Usage

- Create near field monitor  **EM Probes - EM field line** before simulation
- Specify Start and End point
- In Parameters - Storage specify frequencies
- In Parameters - Calculation select field, components, operations
- After simulation switch to **2D Results** Tab
- Select **Plot Type e,h path**

Remarks

- Due to averaging some surrounding cells will be used in the recording

29.1.16. Near field recording

Usage

- Create Field Monitor 
  - Plane, use a single plane for recording
  - Complete Volume, record the complete simulation domain
  - Partial Volume, record a volume specified by a box
- In **Storage Options**, enter frequency points in Hz or time points in time step unit
- For results see Near field animation, Section 29.1.14, or Near field plot, Section 29.1.15

Remarks

- Recording of volumes requires the memory $M = 48 N_x N_y N_z N_f$
- After simulation a file named emvolume.dbx is generated (binary file)

Examples

- Load **Tutorials - Low Pass Microstrip Filter**

29.1.17. Optimization

Usage

- Define variables for geometry, physical or functional properties
- Check variable range and step
- **Simulation** Tab - **Optimization Setup**, Section 24.9
- Click **Create Optimization**
- Add Sources and Goals
- optionally, select hosts in **Host Setup**
- Click **Start**

Remarks

- Several optimizers are available to find the optimum variable set
- The deviation from the goal function is printed in the status window

Examples

- Load **Examples - Coupler & Filters - BP Filter Optimizer**

29.1.18. Parameter Sweep

Usage

- Define variables for geometry, physical or functional properties
- Check variable range and step
- **Simulation** Tab - **Parameter Sweep Setup**, Section 24.7
- Click **Create Parameter Sweep**
- optionally, select hosts in **Host Setup**
- Click **Start**

Remarks

- Sweep independent keeps all but the current variables fixed
- Sweep Parameter space simulates all possible variable combinations

Examples

- Load **Tutorials - 2D - 05 Conformal Patch Antenna**

29.1.19. Phase center calculation (Antennas)

Usage

- Create a Far field monitor, Section 29.1.8
- In Field Monitor - Processing Options, activate **Phase Center Calculation**
- Select phase reference
- Start simulation 

Remarks

- In the log window (and stored in each nf2ff.log file) the phase center is displayed for each far field transformation

29.1.20. Plane wave excitation

Usage

- Click **Create Source** and select **Volume - Plane Wave**
- Enter box coordinates (plane wave is excited inside the box volume)
- Click **Edit Settings**, enter polarization and direction using spherical coordinates

Remarks

- Outside the plane wave box only the scattered field is present
- The plane wave box may not intersect with objects

Examples

- Load **Examples - SAR & ACD - Monkey exposed to plane wave**

29.1.21. Port creation (absorbing)

Usage

- Create a new group for the port (optional)
- Select drawing plane or grid (xy, yz, xz)
- Press **Create Source** and select a line port type, Chapter 20
- Choose perpendicular and propagation direction
- Enter start (open boundary) and end point (feed point)
- Click **Edit Settings** to adjust number, excitation, load, weight, delay, ...
- Set **Port type = Absorbing**

Remarks

- Absorbing ports may only be defined at a boundary with open boundary conditions

Examples

- Load **Templates - Microstrip Line - Impedance**

29.1.22. Port creation (concentrated)

Usage

- Create a new group for the port (optional)
- Select drawing plane or grid (xy, yz, xz)
- Click **Create Source** and select a port type, Chapter 20
- Choose perpendicular and propagation direction
  - Line ports: Enter start (load) and end point (feed) (avoid boundaries)
  - Lumped port: Enter start (+) and end (-) point, connecting conductors
  - Perpendicular lumped port: Enter cross section, top (+) and bottom (-) connecting conductors
  - Click **Edit Settings** to adjust number, excitation, load, weight, delay, ...

Remarks

- Concentrated feed line ports may not be placed at a boundary
- Start or End points of Lumped ports may be placed at a PEC boundary

Examples

- Load **Tutorials - Patch Antenna**
- Load **Tutorials - Low Pass Microstrip Filter**
- Load **Tutorials - Inverted-F Antenna**

29.1.23. Quality factor calculation (Inductance)

Usage

- Create a lumped port (number 1), Section 29.1.22, between a inductive structure, e.g. a coil
- Select **Simulation** tab - **Simulation Config - Equations**
- Right click on sub-1 select **Add Equation - Symbolic Equation**, select equation **ZQ** from the **Template**
- Start Simulation 
- **2D Results**, Plot Type **Impedance** after postprocessing

Remarks

- The quality factor is derived from the input impedance Q=Re(Z1.in)/Im(Z1.in)

Examples

- Load **Tutorials - Multi Layer Spiral Inductor**

29.1.24. RCS calculation

The Radar Cross Section (RCS) of an object can be calculated from the reflected far field of a plane wave excitation.

Usage

- Create a Plane wave excitation, Section 29.1.20
  - the source area should fully enclose the scattering object
  - specify the incident wave direction (in spherical coordiantes)
  - choose polarization (in spherical components)
  - 1 V/m has to be used as field strength
- Create a Far field recording monitor (Boundary), Section 29.1.8. In **Processing Setup** set
  - **Sweep mode - Backscattering**
  - **Normalization - Excitation**
  - **Advanced:** Checkmark **Radar Cross Section (RCS) Calculation**
- After Simulation the result can be plotted in **2D Results** using **Plot Type = Scattering Cross Section**

Remarks

- The RCS result unit is square meters
- The plane wave should not intersect with the object. Some cells space should be betwen plane wave area and object
- Symmetry planes (PEC or PMC) are not supported
- Because the Normalization is set to "Excitation" the result depends on the input power. Therefore the field strength has to be set to 1 V/m

Examples

- Load **Tutorials - Radar Cross Section of a Sphere**

29.1.25. SAR calculation

Usage

- Define excitation scene (antenna, plane wave, ...)
- Create objects with mass densities or use a voxel file, see Chapter 32
- Create a near field recording, Section 29.1.16, covering a volume large enough for mass averaging, minimum 1 cell thickness
- Add a near field animation and specify settings (target frequency, interpolation, averaging)
- Run the simulation 
- Swith to **3D** Tab

Remarks

- Check settings in the **Plot Options** definition (e.g. component SAR_10 for 10g mass averaging)

Examples

- Load **Examples - SAR & ACD - Monkey exposed to plane wave**

29.1.26. S-Parameter calculation

Usage

- Create Port(s), Section 29.1.22
- Enter unique port numbers for multiple ports
- Switch on/off port excitation (minimum 1 port excited)
- Start Simulation
- Results in **2D Results** Tab, **Plot Type = Scattering Parameters**

Remarks

- Each excited port with unique number n creates a subfolder sub-n
- In folder sub-n the results sn_k are available after postprocessing

Examples

- Load **Tutorials - Patch Antenna**
- Load **Tutorials - Low Pass Microstrip Filter**
- Load **Tutorials - Inverted-F Antenna**

29.1.27. Touchstone file (postprocessing)

Usage

- All ports for the scattering matrix have to be excited
- In **Simulation Setup** select **Simulation Mode** = Sequential Excitation & Y-Parameter Extraction

Remarks

- The Touchstone file (e.g. `default.s2p`) will be created in the **ypar** folder
- Results in the **ypar** folder are derived from Y-Parameters and are independent from the port impedance

Examples

- Load **Tutorials - Chip Package on PCB**

29.1.28. User equations (postprocessing)

Usage

- Select **Simulation** tab
- Click **Simulation Config - Equations**
- Right click on desintation folder, select **Add Equation**
  - **Symbolic Equation** Apply user defined formulas on result files
  - **Calculation** Predefined functions (e.g. maximum search)
  - **File Copy** interchange between subfolders, file abbreviations
- Enter a unique **Destination** Name
- Add **Source** files if applicable
- Enter equation in the **TERM** window or select a template
- Run post processing

Remarks

- Please refer to www.python.org for valid mathematical expressions
- Destination and source files require to have the same format (i.e. frequency points, magnitude, phase)
- Note that all frequency domain results are complex
- The result file will be located in the selected subfolder. Use the **import** button in the plot setup to load and display the graph

Examples

- Load **Tutorials - Multi Layer Spiral Inductor**

29.1.29. Waveguide excitation (hollow)

Usage

- Create Waveguide Port(s), Chapter 20
- Select **Port Parameters - Start Port Definition**
- **Numbers of Start Port** - Enter number of modes to be used for decomposition at this port, starting with the next free port number
- **Start Port Excitation** - Specify which mode should be excited using one of the port number above

Remarks

- Magnetic walls are not supported in waveguide calculation. For exploiting symmetry, only electric walls may be used
- The number range in the waveguide port definition specifies the number of modes which are taken into account in the simulation for this waveguide beginning with the mode having the lowest cut-off frequency
- The program calculates the modes just before the main simulation starts. They are listed as TE0, TE1, TE2, ... TM0, TM1, TM2, ...
- Their cut-off frequencies are listed in the log window together with their polarizations.

Example The numbers in the waveguide port definition specify port start number and the number of modes which are taken into account in the simulation, i.e. 2-4 means that the first 3 modes are used and port number 2 is assigned to the mode which has the lowest cut-off frequency.

The first modes of a circular waveguide are:

- 1st TE_{11} pol1
- 2nd TE_{11} pol2
- 3rd TM_{01}
- 4th TE_{21} pol1
- 5th TE_{21} pol2
- 6th TM_{11} pol1
- 7th TM_{11} pol2
- 8th TE_{01}
- ...

Some modes have 2 polarizations. With port numbers 2-4 the program determines 3 TE modes first, e.g.

- TE0, 10 GHz, x-pol, (TE_{11})
- TE1, 10 GHz, y-pol, (TE_{11})
- TE2, 16.6 GHz, x-pol, (TE_{21})

Then 3 TM modes are determined, e.g.

- TM0, 13.1 GHz, (TM_{01})
- TM1, 20.8 GHz, x-pol, (TM_{11})
- TM2, 20.8 GHz, y-pol, (TM_{11})

Now they will be sorted according to their cut-off frequency:

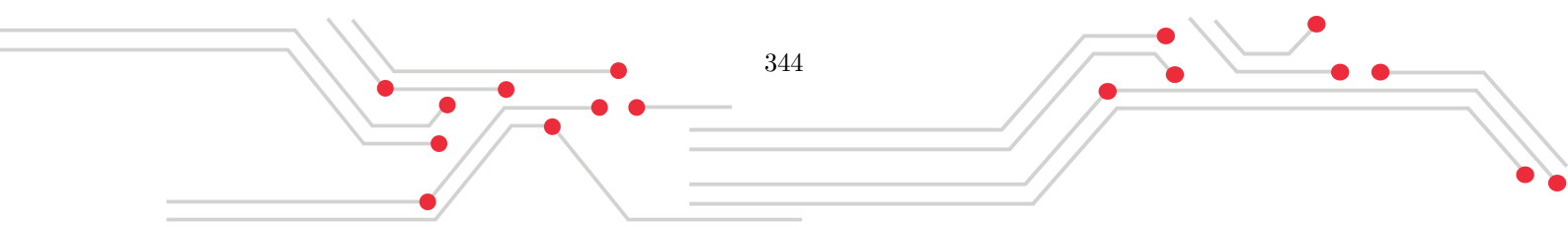
- Mode 1 (Port 2) = TE0 (TE_{11})
- Mode 2 (Port 3) = TE1 (TE_{11})
- Mode 3 (Port 4) = TM0 (TM_{01})

This means: If the mode TM_{01} should be excited, at least 3 modes have to be specified in the port set up

- Numbers of start port 2-4
- Start port excitation 4

Examples

- Load **Tutorials - Reflector Antenna with feed**
- Load **Examples - Waveguide Devices - Circular Waveguide Branch**



29.2. Application Notes

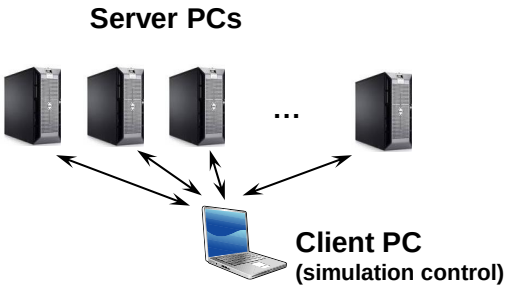
29.2.1. Batch Processing

EMPIRE User Guides
Applications
Releases

Batch Processing

OBJECTIVES

- To execute several jobs after another, the batch control is available

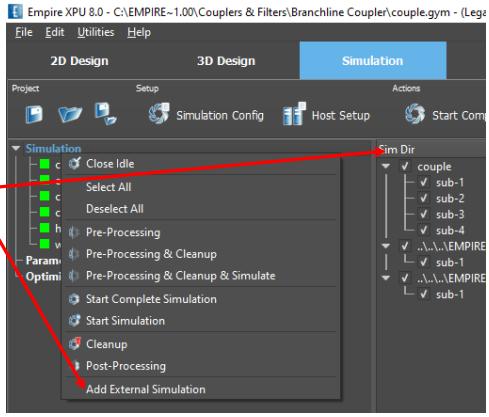


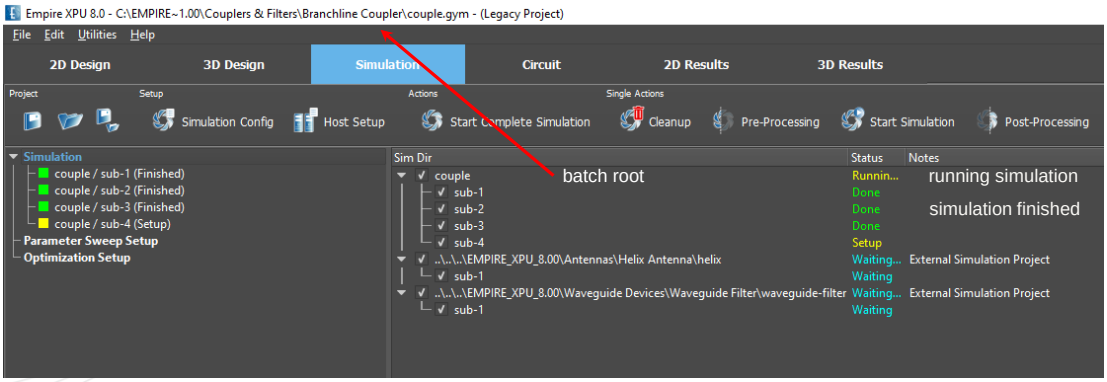
GENERAL REQUIREMENTS

- Client PC (e.g. Intel Core i7)
- Optional Server PCs for Simulation
- EMPIRE XPU™ Silver Package on client and servers

USAGE

- Setup all single simulations on local PC (.acad file must exist in the folder trees)
- Right Click on “Simulation” – “Add External Simulation” to add simulation folder
- Select .acad file
- Select jobs for simulation
- All selected jobs will be simulated on after another on the defined hosts in the Hosts Setup”
- Local host can always be used, with remote control feature all Servers with Empire in the network can be used (see AN-remote.pdf)





EMPIRE XPU™ is a product of IMST GmbH, Carl-Friedrich-Gauss-Str. 2, 47475 Kamp-Lintfort, Germany
P +49 2842 981-400, F +49 2842 981-499, E empire@imst.de, I www.empire.de

29.2.2. CAD Import

User Guides
EMPIRE Applications
Releases

CAD Import



OBJECTIVES

- Import CAD data from third party software into Empire XPU
- Select customized options for current usecase during import

REQUIREMENTS

- External CAD data in supported file format (see list)
- Empire XPU Import License for this file format

PROCEDURE

1. Import Procedure
- Press File – Import – 3D Solids – Import *
- Select file (only cad files with selected format will be shown)

2. Import Procedure
- Press Import Button in Main menu
- Select any file (all supported cad files will be shown)

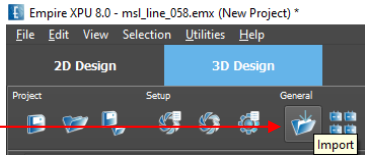
CAD CONVERTER OPTIONS

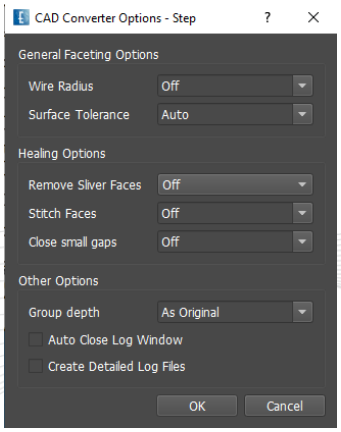
If import needs a CAD conversion the CAD Converter Options window shows up

- **Wire Radius:** Option for import of B-Rep wires, (Off: no wire import, value: radius of imported wires)
- **Surface Tolerance:** Amount of geometric details resulting from import. The given value is the maximum allowed deviation between the original surface and import result
- **Healing Options:** Different settings for CAD data repair *
- **Group depth:** Settings for group structure after the import *

SUPPORTED FORMATS

3DXML: .3dxml
ACIS: .sat, .sab, .asat, .asab
CATIA V4: .model, .exp, .session
CATIA V5: .CATPart, .CATProduct, .CGR
CATIA V6: .CATPart, .CATProduct, .CGR, .3DXML
DXF/DWG: .dxf, .dwg
IGES: .igs, .iges
Inventor: .ipt,
JT: .jt
Siemens NX: .prt
Parasolid: x_t, .xmt_txt, .x_b, .xmt_bin, .p_b, .xmp_bin, .p_t, .xmp_txt
Pro/E / Creo: .prt, .prt.*, .asm, .asm.*
Solid Edge: .par, .asm, .psm
Solid Works: .sldprt, .sldasm
STEP: .stp, .step
Nastran: .nas





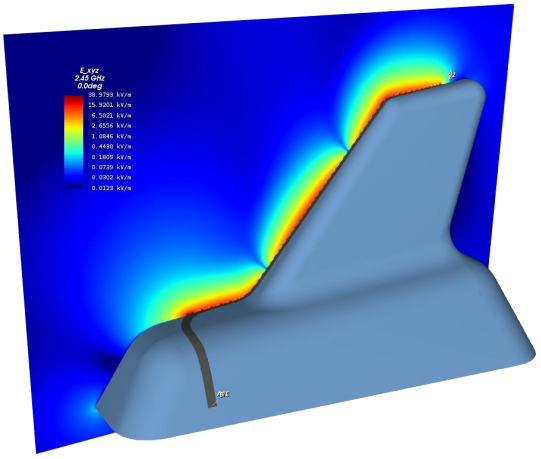
** More information about this settings in CAD Data Import Tutorial and Empire Manual*

EMPIRE XPU™ is a product of IMST GmbH, Carl-Friedrich-Gauss-Str. 2, 47475 Kamp-Lintfort, Germany
P +49 2842 981-400, F +49 2842 981-499, E empire@imst.de, I www.empire.de

29.2.3. Conformal Toolbox

User Guides
EMPIRE Applications
Releases

Conformal Toolbox



OBJECTIVES

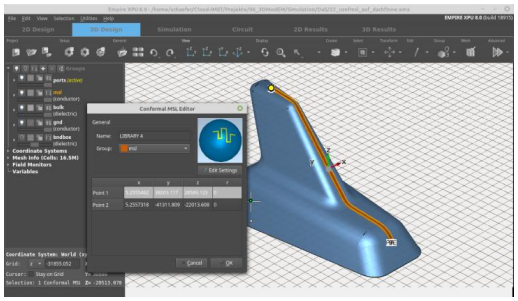
- Modeling of non-planar HF structures on arbitrary 3D surfaces
- Molded Interconnect Device (MID) design

REQUIREMENTS

- License for EMPIRE 8.2 or higher

FEATURES

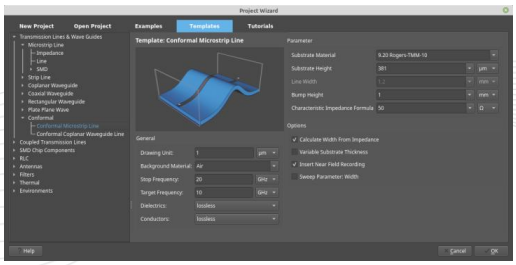
- Versatile conformal library elements (LEs)
  - Can be “painted” onto arbitrary 3D surfaces
  - Based on reference points
  - Different types
    - Conformal MSL
    - Conformal CPW
    - Conformal Viafence
    - Conformal Part (projected 2D-layout)
  - Advanced LE features, e.g.
    - Automatic width for MSL to compensate for variable substrate thickness
    - Support for buried CPW
    - Optional CPW back metallization with integrated viafence for odd-mode cancelation
    - Smart 1-click automesh options



- Conformal project templates
  - Make getting started very easy
  - Parametrized geometry calculated from given characteristic impedance



- New “Metal Sheet Stack” material
  - Efficient modeling of thin metal coatings
  - Advanced roughness modeling based on Gold Helmreich approach
  - Surface- and inter-sheet roughness selectable



EMPIRE XPU™ is a product of IMST GmbH, Carl-Friedrich-Gauss-Str. 2, 47475 Kamp-Lintfort, Germany
P +49 2842 981-400, F +49 2842 981-499, E empire@imst.de, I www.empire.de

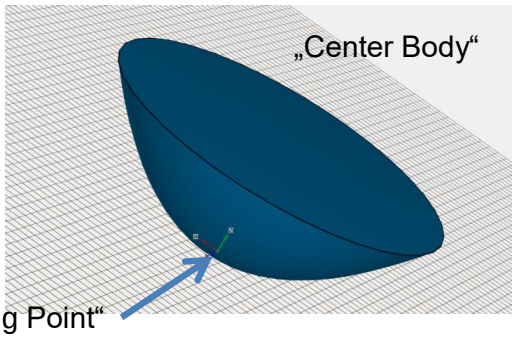
29.2.4. Conformal-Dielectric

User Guides
EMPIRE Applications
Releases

Conformal Dielectric

OBJECTIVES

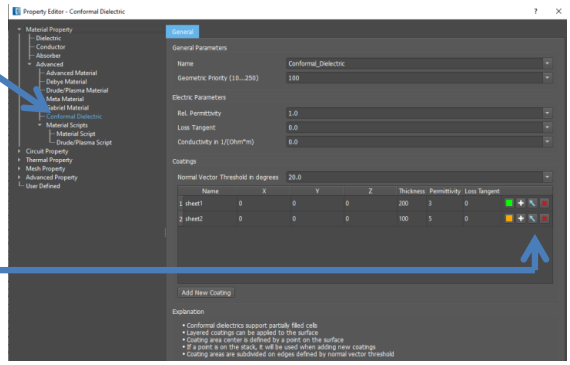
- Apply a layered dielectric coating to an arbitrarily curved surface
- Take into account thin dielectric sheets without resolving the sheet thickness with fine mesh cells
- Calculate cell coefficients from layered dielectrics based on local Finite Element Method



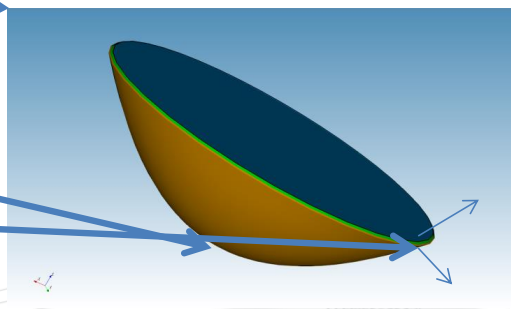
In Empire a solid body is a closed surface constructed from small triangles

PROCEDURE

1. Define Conformal Dielectric
- Create a group for the coated object
- Assign “Conformal Dielectric” property to group
- Set permittivity, conductivity and loss tangent for the center body
- Click Add New Coating
  - set thickness, permittivity, loss tangent for the first layer
  - “Pick Coating Point” (X,Y,Z on surface) where the coating should be applied in the vicinity (click on surface in 3D Design Mode)
  - Optionally continue with next coating



2. Verify Coating
- Switch to 3D Results Mode



Notes

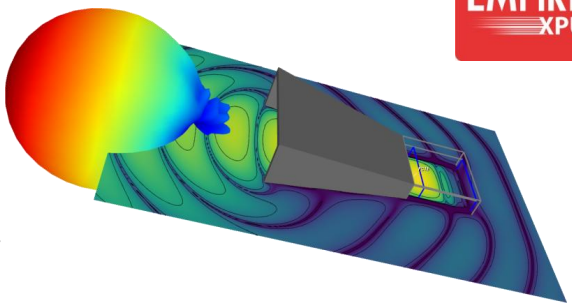
- Normal vector threshold is used to identify edges of piecewise smooth surfaces
- Coating areas are extended from “Coating Point” to edges (if normal angle difference of neighbor triangles larger than threshold)
- Several Conformal Dielectric objects may not intersect
- Thermal simulations are not supported

EMPIRE XPU ™ is a product of IMST GmbH, Carl-Friedrich-Gauss-Str. 2, 47475 Kamp-Lintfort, Germany
P +49 2842 981-400, F +49 2842 981-499, E empire@imst.de, I www.empire.de

29.2.5. CST Import

User Guides
EMPIRE Applications
Releases

CST Import



OBJECTIVES

- Project migration from CST to Empire XPU
- Minimization of user effort by automatically transferring simulation settings

REQUIREMENTS

- CST project file (.cst)
- CST project folder, including:
  - /Model/3D/Model.mod
  - /Model/3D/Model.par (optional)
  - /Result/Model.msh (optional)
- Geometry exported as STL (.stl) from within CST. Must be placed beside project file

PRESERVED FEATURES

- Geometry
  - Grouping of components (similar to CST)
  - Colors of objects
- General settings, e.g. drawing unit, frequency range, target frequency, boundary conditions
- Materials
  - Background material
  - Parameter based materials
- Lumped elements, e.g. SMD Resistors, Capacitors, Inductors
- Excitation
  - “DiscretePort” and “DiscreteFacePort”
  - Placeholder for incompatible Ports
- Mesh
- Far-field monitors

PROCEDURE

- Click “File – Import – Projects – Import CST”
- Select .cst file

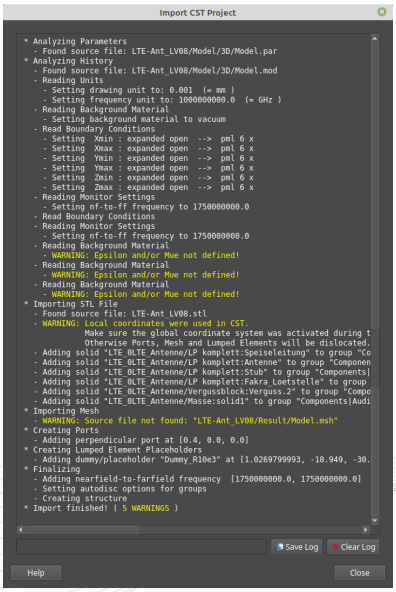
Alternative Import Procedure:

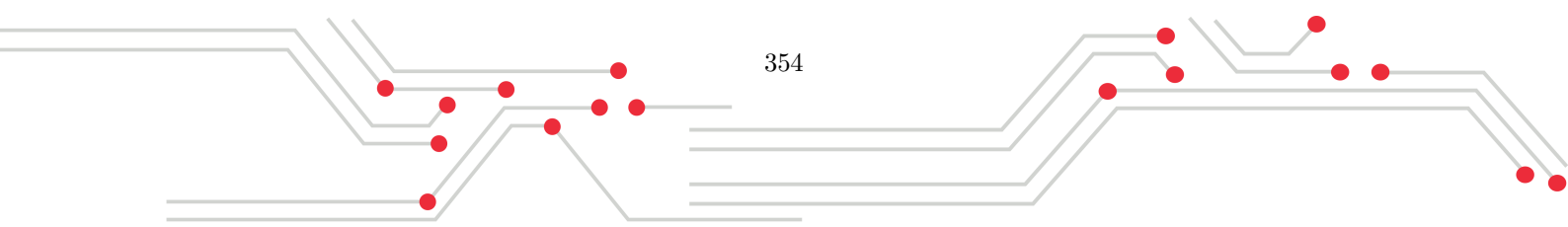
- Click the Import Button in Main menu
- Select any file (all supported cad files will be shown, including .cst files)



TIPS AND TRICKS

- Best import result can be expected if:
  - No local coordinate systems (LCS) are being used in CST project
  - Only ports of type “DiscretePort” or “DiscreteFacePort” are being used
  - CST project was created “straight forward” without a lot of copy&paste
- Import is based on CST project history, so re-editing problematic details within CST and saving again might help.
- Enable world coordinate system (WCS) during STL export from CST
- View import log file for possible errors and warnings:





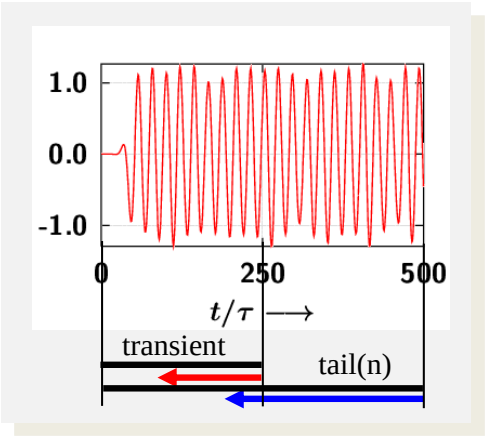
29.2.6. Signal Estimation

User Guides
EMPIRE Applications
Releases

Resonance Estimation

Objectives

- Especially for high-quality structures the usual end criteria of decaying energy will either lead to very long simulation times or to truncation errors. In those cases a smart resonance estimation technique based on a moving averaging algorithm can be applied to reduce simulation times drastically



Time series from a dielectric resonator

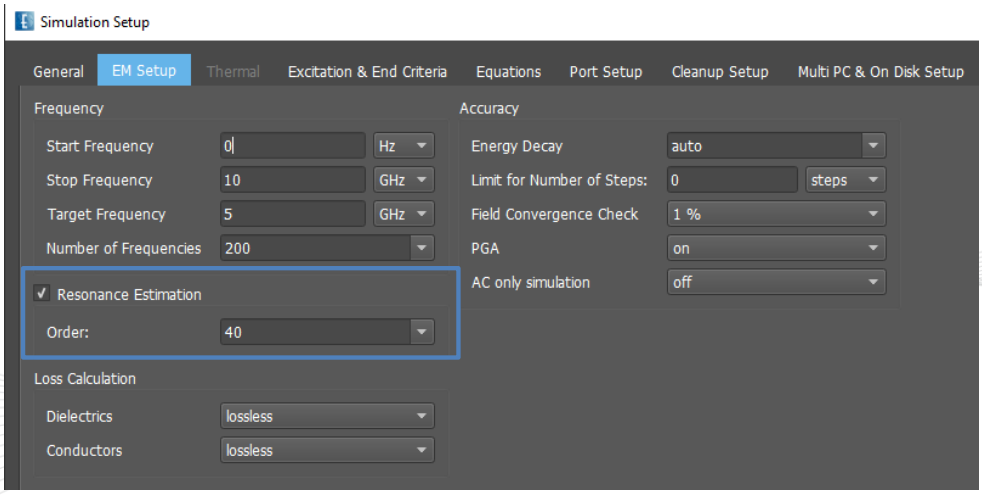
Principle

- The begin of the time series is treated with the standard DFT
- A cybernetic model system is calculated from TD data, but evaluated in the Frequency Domain
- A resonance estimation with the order of n needs about 2*n sampled time domain values after the transient part of the signal
- An estimation order of 40 achieves good results for most cases; for very high-quality structures and structures with lots of resonances it is recommended to increase the estimation order
- In order to access the order set “Structure Type” = “User Setup” in General Tab
- A subsequent Postprocessing applies the new order

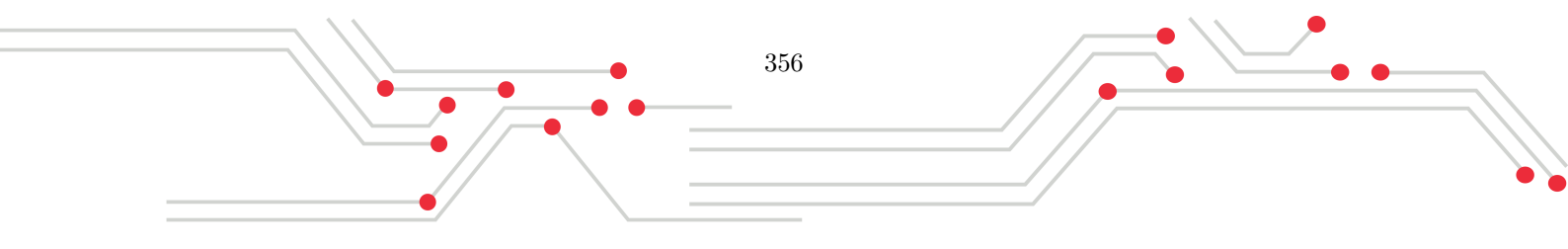
$$S(\omega) = \sum_{k=0}^{M} \mathrm{e}^{-\mathrm{j}\omega\Delta k} s_k + z^{-M-1} S_{\mathrm{m}}(z)\big|_{z=\mathrm{j}\omega\Delta}$$

transient tail

DFT combined with resonace estimation



EMPIRE XPU™ is a product of IMST GmbH, Carl-Friedrich-Gauss-Str. 2, 47475 Kamp-Lintfort, Germany
P +49 2842 981-400, F +49 2842 981-499, E empire@imst.de, I www.empire.de



29.2.7. Far-Field Recording

User Guides
EMPIRE Applications
Releases

FarField Calculation

OBJECTIVES

- Calculate far field radiation patterns (Eabs, Ephi, Etheta, ELHCP, ERHCP, Axial Ratio)
- Calculate Efficiencies, beamwidth, beam angle.

REQUIREMENTS

- Field Monitor: EM Farfield
- Open boundary conditions
- Space to boundary > lambda/8

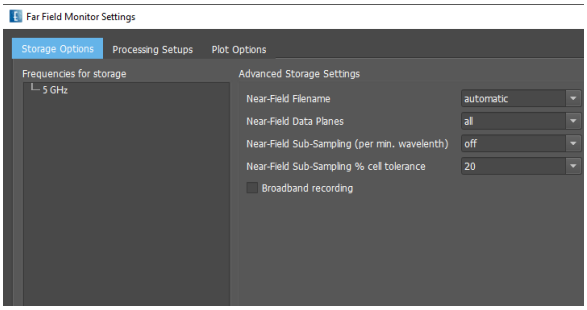
PROCEDURE

1. Create FarField Monitor
2. In “Storage Options”, define frequencies
3. In “Processing Setups” - Far Field Setup 1,2,…,6 define transformation parameters
4. In section “Plot Options” set animation properties for 3D pattern display
5. Display Origin is used as the center for display

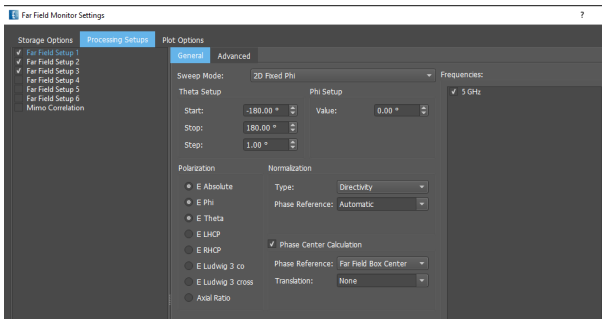
PARAMETERS

- Normalization (Directivity, Gain, Gain_Real, Excitation,…)
- Sweep Mode - Defines 2D or 3D patterns
- Angle Setup - Start Stop Step
- Polarization – Define components to be calculated

Farfield Storage Parameters



Farfield Transformation Parameters



Result files naming convention

Nearfield on Huygens Surface (binary):
`Farfield_1`
3D Farfield (ASCII):
`Farfield_1_f5.00000e+09_ephi`
2D Farfield (ASCII):
`Farfield_1_f5.00000e+09_p0.000e+00_ephi`
1D Farfield (ASCII):
`Farfield_1_f5.00000e+09_p0.000e+00_t9.000e+01_ephi`

f : frequency in Hz
p: phi angle in degrees
t : theta angle in degrees
.log files for each far field file contain efficiencies and further results

EMPIRE XPU™ is a product of IMST GmbH, Carl-Friedrich-Gauss-Str. 2, 47475 Kamp-Lintfort, Germany
P +49 2842 981-400, F +49 2842 981-499, E empire@imst.de, I www.empire.de

29.2.8. Far-Field Display

User Guides
EMPIRE Applications
Releases

Far Field Visualization

OBJECTIVES

- Display 2D far field patterns in linear or polar plot
- Display 3D far field patterns together with structure

REQUIREMENTS

- Field Monitor: EM Farfield
- 2D pattern: Fixed angle in Sweep Mode
- 3D pattern: Sweep both in Sweep Mode

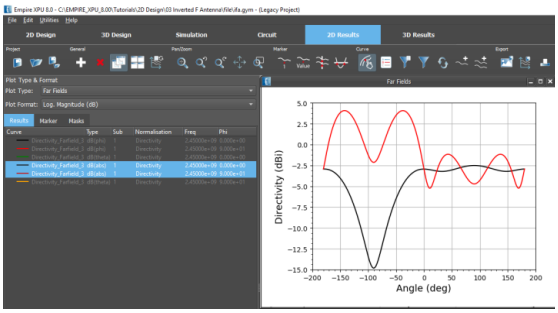
PROCEDURE

1. Create Far Field Monitor
2. Simulation and Far Field Transformation in Postprocessing (enabled by default)
3. 2D Result: Plot Type – Far Fields, Plot Format – Lin or Log, Polar
4. 3D Result: Switch on far field monitor, right click to edit, adjust settings in Plot Options

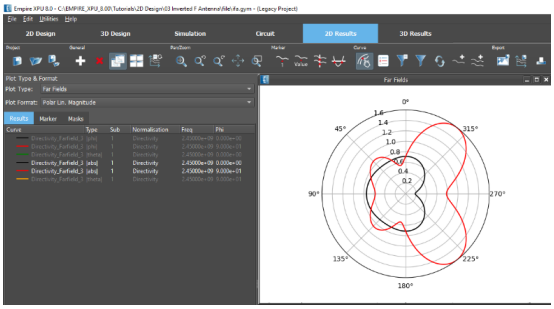
DISPLAY PARAMETERS

- Data Source - If set to automatic a 3D pattern is selected, 2D pattern can be displayed, too. Frequency and component selection
- Scaling - Defines the linear or logarithmic range for amplitude and color. Range and max value
- Display – Size, map, style of plot, legend entries

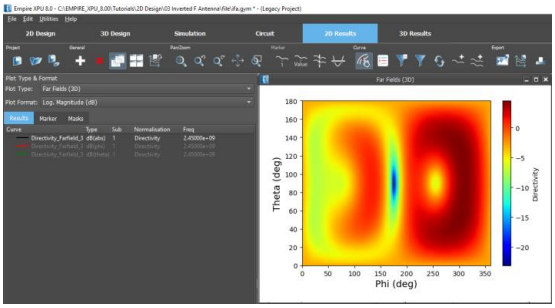
2D Result: Farfield Linear Plot



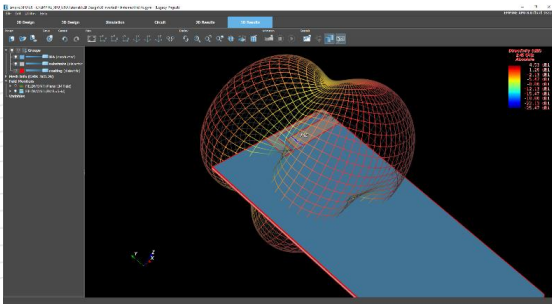
2D Result: Farfield Polar Plot



2D Result: Far Field (3D)



3D Result: Far Field



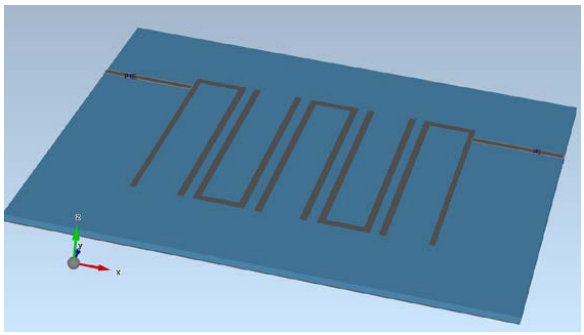
29.2.9. Filter Template

EMPIRE User Guides
Applications
Releases

Filter Synthesis and full wave analysis

OBJECTIVES

- Filter design in microstrip or stripline technology
- Coupled line, coupled hairpin or tapped hairpin approach
- Selectable material, Chebycheff order, resonance frequency , ripple and impedance
- Synthesis by built-in formulas, analysis by 3D FDTD method

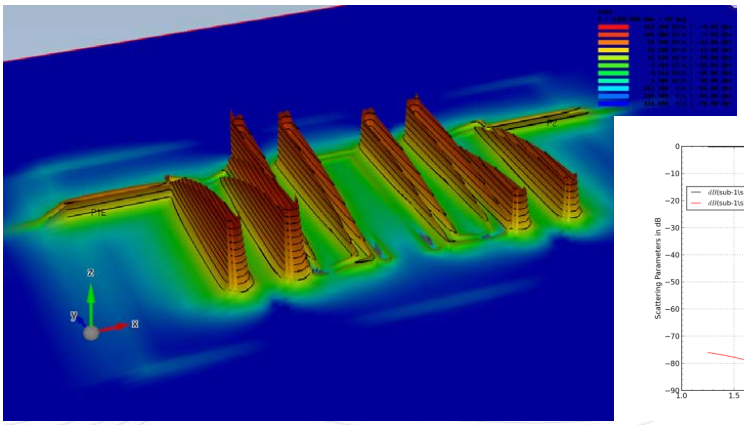


GENERAL REQUIREMENTS

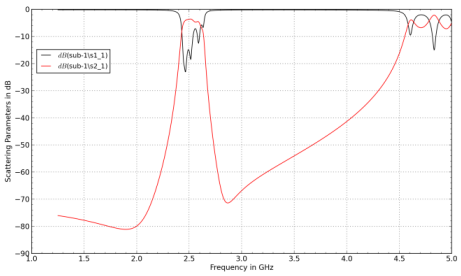
- Windows or Linux PC (e.g. Intel Core i5)
- EMPIRE XPU™ Silver Package
- Demo Version available at www.empire.de

USAGE

- Click “Templates” - “Filters” at program start
- Select Filter Type
- Select Substrate Material and thickness
- Enter Filter order, resonance frequency, bandwidth and ripple level
- Select feed line and filter impedance
- Optionally, set coupling length as sweep parameter
- Click Open Selection
- Click Simulation Control - Simulation



Magnitude of electric field at 2.45 GHz
and S-Paramaters

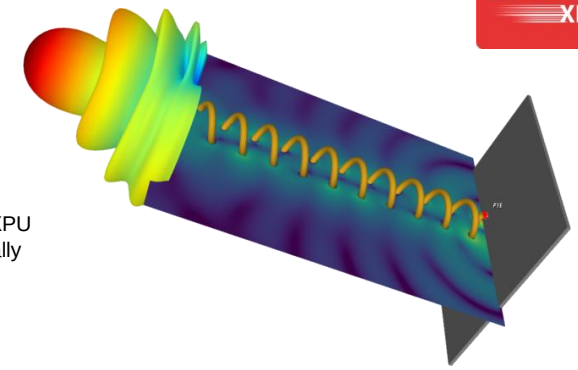


EMPIRE XPU™ is a product of IMST GmbH, Carl-Friedrich-Gauss-Str. 2, 47475 Kamp-Lintfort, Germany
P +49 2842 981-400, F +49 2842 981-499, E empire@imst.de, I www.empire.de

29.2.10. HFSS Import

User Guides
EMPIRE Applications
Releases

HFSS/AEDT Import



OBJECTIVES

- Project migration from AEDT to Empire XPU
- Minimization of user effort by automatically transferring simulation settings

REQUIREMENTS

- HFSS/AEDT project file (.hfss, .aedt)
- Geometry exported in ACIS format (.sat, .sab or .sm3) from within AEDT. Must be placed beside project file
- Import license for ACIS files

IMPORT HIGHLIGHTS

- Support of projects with multiple:
  - Models
  - Solver setups
  - Sweep configurations
- Preserves the following features during import:
  - Geometry including object color
  - General settings, e.g. drawing unit, frequency range, target frequency, boundary conditions
  - Materials
    - Parameter based materials
    - Anisotropic materials
    - Sheet materials
  - Lumped ports with integration line

PROCEDURE

- Click “File – Import – Projects – Import HFSS/AEDT”
- Select .hfss or .aedt file

Alternative Import Procedure:

- Click the Import Button in Main menu 
- Select any file (all supported cad files will be shown, including .hfss and .aedt files)

TIPS AND TRICKS

- Best import result can be expected if only ports of type “Lumped Port” are being used
- In case of incompatible ports, placeholder will be created to aid manual reconstruction
- Try different settings for “Surface Tolerance” during geometry import
- View import log file for possible errors and warnings:



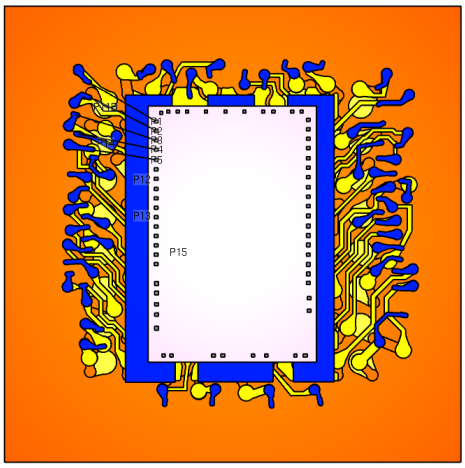
29.2.11. Layout Export

User Guides
EMPIRE Applications
Releases

Layout Export

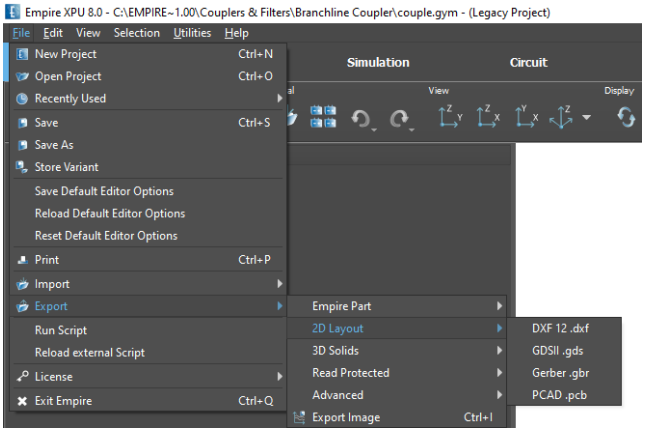
OBJECTIVES

- Export layout data of printed circuits from Empire XPU
- Different export formats available:
- GDSII, GERBER, PCAD, 2D DXF



PROCEDURE

- Start Empire XPU
- Open project to be exported
- Switch to 2D Design – choose view
- Press File – Export – 2D Layout
- Choose export file format
- Select path and filename



REQUIREMENTS

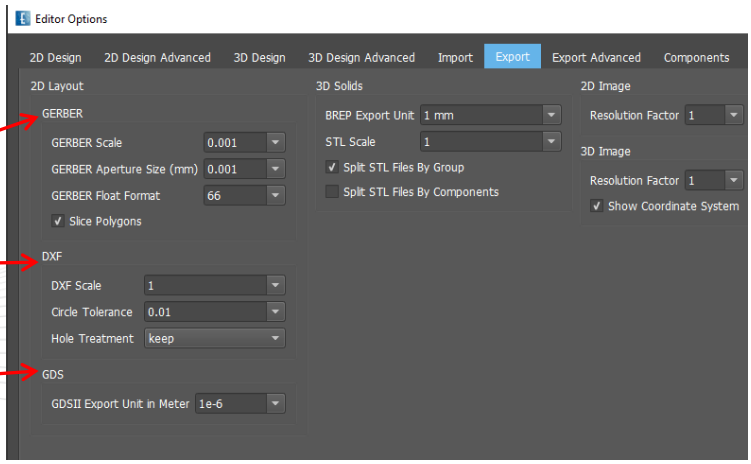
- Input file with layout data
- GDS: Group names have to be numbers
- Export varies for different view planes

Options for Export:
Editor Options - Export

Gerber Export parameters

DXF Export parameters

GDS Export parameters

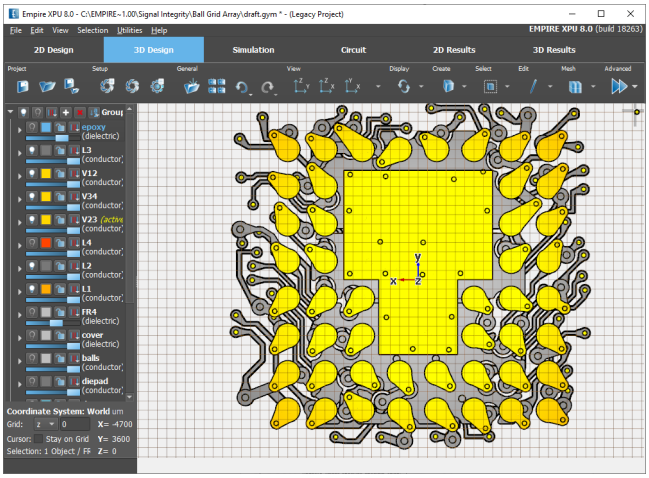


EMPIRE XPU™ is a product of IMST GmbH, Carl-Friedrich-Gauss-Str. 2, 47475 Kamp-Lintfort, Germany
P +49 2842 981-400, F +49 2842 981-499, E empire@imst.de, I www.empire.de

29.2.12. Layout Import

User Guides
EMPIRE Applications
Releases

Layout Import



OBJECTIVES

- Import layout data of printed circuits into Empire XPU
- Assign extrusion heights of layers
- Set priority numbers and material properties

REQUIREMENTS

- External layout data in supported file format (see list)
- ODB ++ import requires license

PROCEDURE

1. Start Empire XPU and change to the folder of the layout data, e.g. test.dxf
2. Select File – Import – 2D Layout – Import *
3. Select file
4. De-select layers optionally,
5. Optionally rename, set properties, assign heights
6. Click Import

SUPPORTED FORMATS

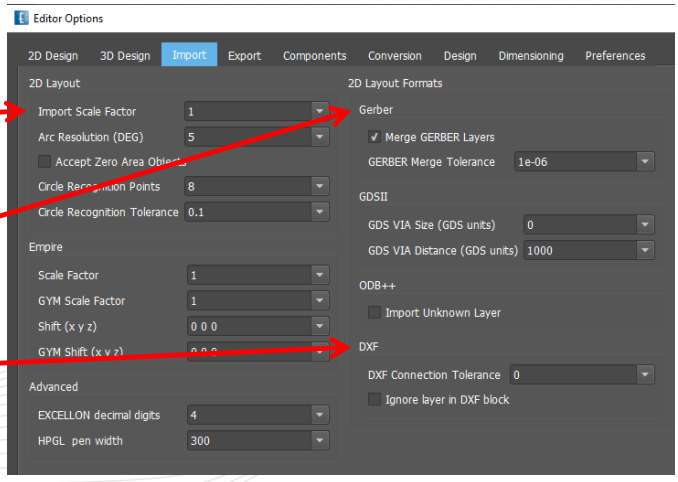
DXF (2D): ..dxf
GDSII: .gds
Gerber: .ger, .gbr, .gbx, .exg, .pho
ODB++: .tgz

IMPORT SETTINGS

- Scale factors, for mil to micron use 1/25.4
- Options to adjust automatic recognition of circles from polygons

Gerber: Options to merge GERBER layers with defined tolorance

DXF: Connects lines to polygons if distance of endpoints is closer than defined connection tolerance



29.2.13. Mesh Setup

User Guides
EMPIRE Applications
Releases

Discretisation

General Remarks

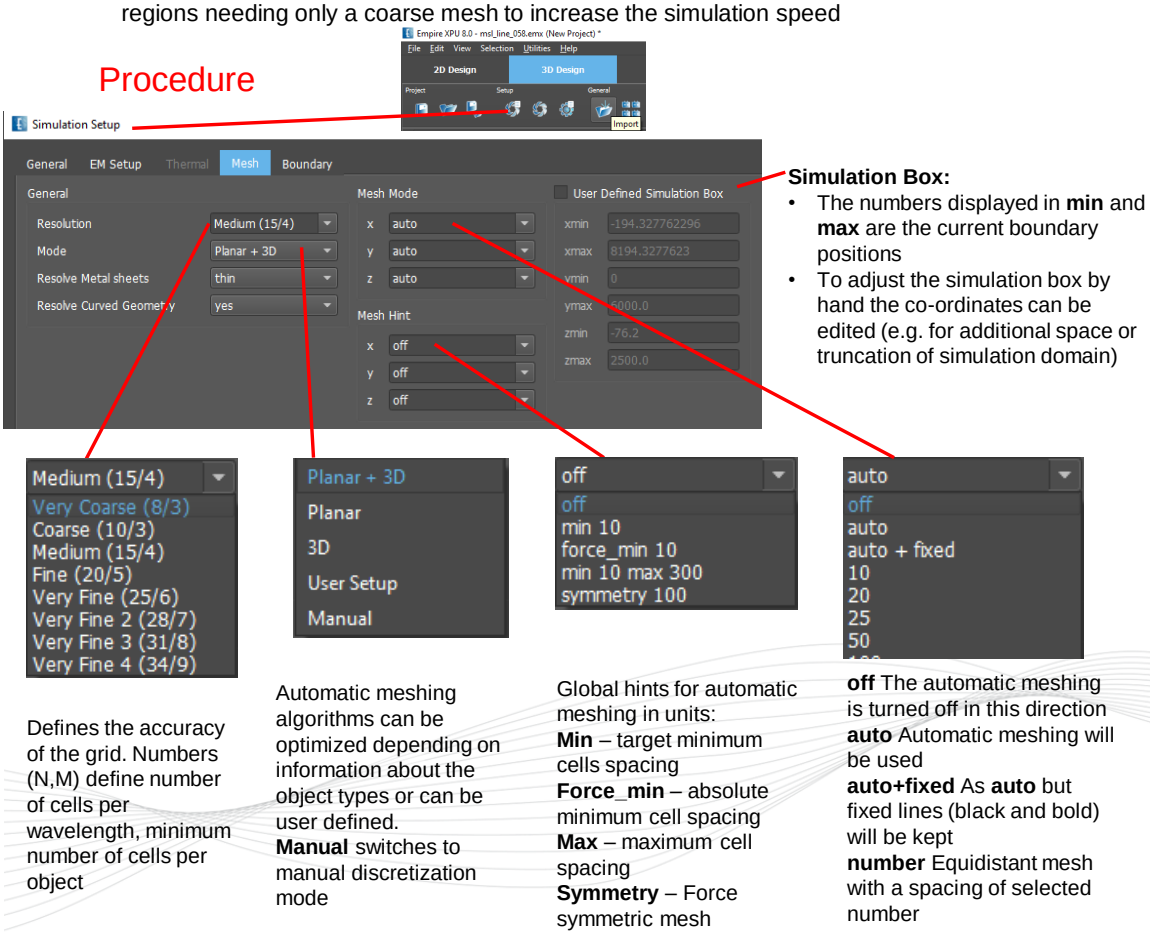
The meshing can be defined in several ways:

- using the *Automatic Meshing*default mode or with user defined settings
- using the **Automatic Meshing** mode with meshhints defined for a group
- **Manual** by selecting a disc-bar or a library element and using the functions from the toolbar

General Rules

- the largest cell size of the mesh should be smaller then the tenth of the smallest wavelength
- the smallest cell determines the time step for the simulation → very small cells lead to long simulation runs. Setting minimum cell size is possible in Automesh Setup - Directions
- a graded mesh is available between regions needing a fine mesh and the regions needing only a coarse mesh to increase the simulation speed

Procedure



Simulation Box:

- The numbers displayed in **min** and **max** are the current boundary positions
- To adjust the simulation box by hand the co-ordinates can be edited (e.g. for additional space or truncation of simulation domain)

Defines the accuracy of the grid. Numbers (N,M) define number of cells per wavelength, minimum number of cells per object

Automatic meshing algorithms can be optimized depending on information about the object types or can be user defined.
Manual switches to manual discretization mode

Global hints for automatic meshing in units:
Min – target minimum cells spacing
Force_min – absolute minimum cell spacing
Max – maximum cell spacing
Symmetry – Force symmetric mesh

off The automatic meshing is turned off in this direction
auto Automatic meshing will be used
auto+fixed As **auto** but fixed lines (black and bold) will be kept
number Equidistant mesh with a spacing of selected number

EMPIRE XPU™ is a product of IMST GmbH, Carl-Friedrich-Gauss-Str. 2, 47475 Kamp-Lintfort, Germany
P +49 2842 981-400, F +49 2842 981-499, E empire@imst.de, I www.empire.de

29.2.14. Multilayer Meshing

User Guides
EMPIRE Applications
Releases

EMPIRE XPU

Efficient meshing strategies for ML-PCB & LTCC design

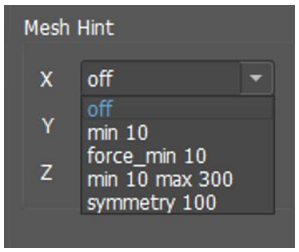
Objectives

- Multilayer PCB and LTCC designs often feature a plethora of small features, which makes them difficult to mesh correctly
- The simple tips described below can help the user make educated guesses at the required mesh resolution
- By defining group-specific and/or global mesh rules, the user can reduce model complexity, increase the time step and, ultimately, reduce simulation time without loss of accuracy

Automatic & Semi-automatic Meshing

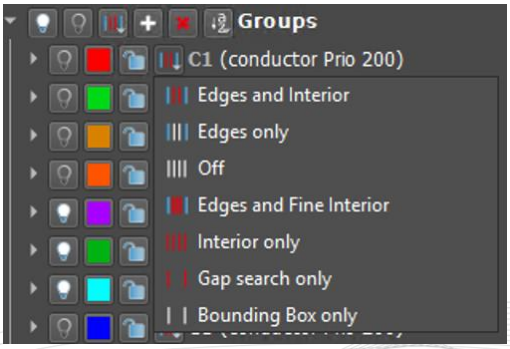
Choice of Mesh Resolution

- Start by using a coarse mesh and refine later to, e.g., medium or fine
- Define a Global mesh hint that limits the smallest mesh size to a reasonable value, e.g., 25 µm
- It is not needed to mesh much finer than the production accuracy (mechanical tolerances)
- A multilayer design often has objects in different layers, whose projection on the xy-plane may yield very small distances
- These small features are, by default, resolved
- Defining a mesh hint with “min 25” or “force min_25” prevents this



Definition of Group-Specific Mesh Options

- The auto-mesher considers by default all objects in a group
- Round via-holes in mutilayer PCB/LTCC designs usually lead to a dense mesh for the via-holes
- This is not needed and, for efficient simulations, it is recommended to set the automeshing mode for the VIA groups to “Edges only”
- If a group has objects that are not important for RF simulation (e.g., solder mask), then this group should be set to “Off”
- If a group contains, e.g., a plastic casing, it is often not needed to resolve small gaps and round edges; the automatic meshing of the group can be set to “Bounding Box only”

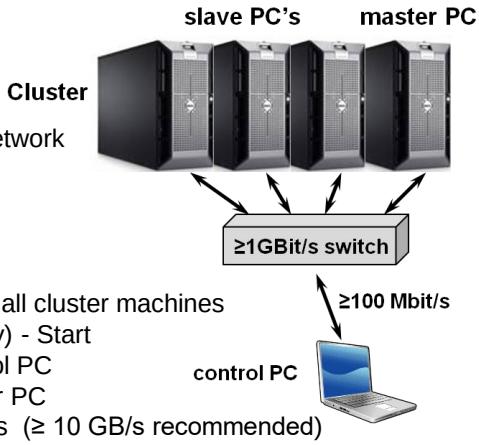


EMPIRE XPU™ is a product of IMST GmbH, Carl-Friedrich-Gauss-Str. 2, 47475 Kamp-Lintfort, Germany
P +49 2842 981-400, F +49 2842 981-499, E empire@imst.de, I www.empire.de

29.2.15. MultiPC Processing

EMPIRE User Guides
Applications
Releases

Multi PC Solver



OBJECTIVES

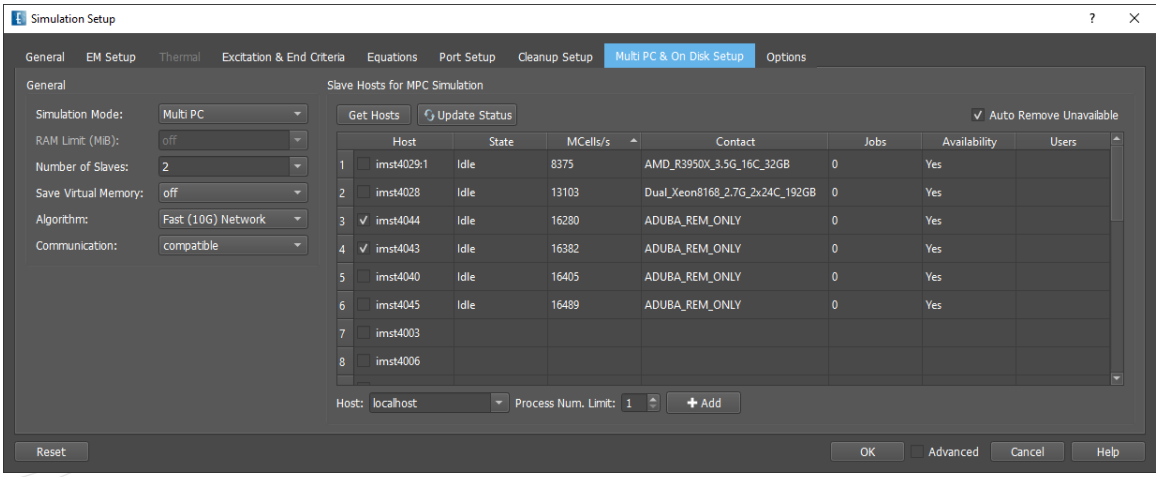
- Simulate one job on multiple PCs within a fast network
- Increase performance by usage of multiple PCs
- Increase memory for large structures

GENERAL REQUIREMENTS

- EMPIRE XPU™ **Legacy Server** to be started on all cluster machines using EMPIRE XPU™ - Utilities – Server (Legacy) - Start
  - Frontend modules needed on the control PC
  - EMP_MP module needed on the master PC
- ≥ 1 GB/s network connection between server PCs (≥ 10 GB/s recommended)
- Firewall exception for ports 42000, 45000-47000
- Hostnames of servers should be known (or IP addresses)
- Same architecture for master and slave PCs
- Administrator User or start Empire with Admin rights

USAGE

- Set up / load model on control PC
- Switch to 'Simulation' Tab – click Simulation Setup - 'Multi PC & On Disc Setup'
- Set "Simulation Mode" = "Multi PC", define the total "Number of Slaves" for the simulation
- Click "Get Hosts" or enter "Host" and "Add", enable the Slave PCs for the simulation
- Optionally select Algorithm, Communication, Close with OK
- Click 'Host Setup' and select the Master PC, Close with OK
- Click Start Complete Simulation



29.2.16. Optimizer

User Guides
EMPIRE Applications
Releases

Optimizer

OBJECTIVES

- variation and optimization of parameterized simulation models

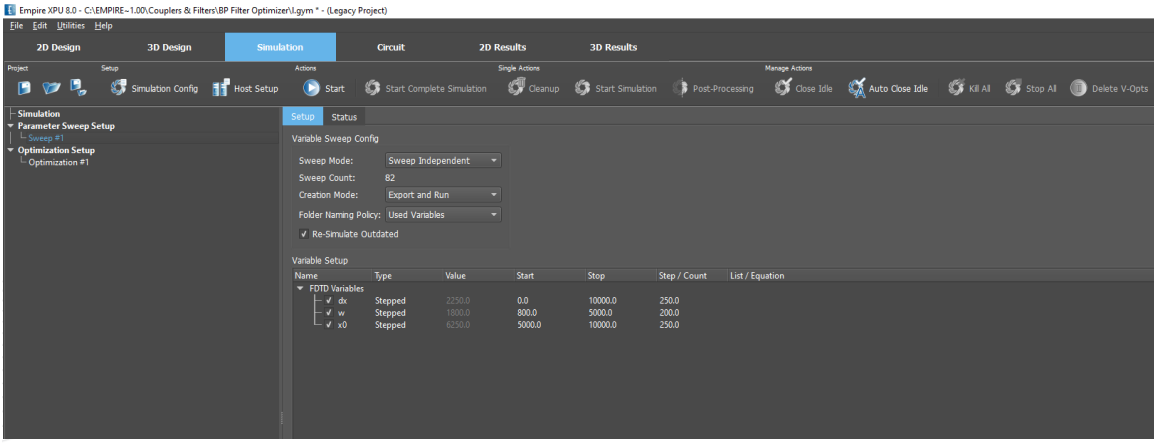
GENERAL PRINCIPLE

- Parameterized Model needed (see Application Note AN-parametric)
- Parameter Sweep: Sweep all parameters within the parameter range and with the parameter step entered in Variables.
- Optimizer: Different Optimizer algorithms to achieve a user defined Optimization GOAL. The optimization begins with the current parameter set entered in the variables. From this point in parameter space it searches for better solutions at neighbor points. The distance from the current point to the neighbor point is given by variable step TIMES the current step factor.
  If one of the neighbors hits the GOAL better, it is taken as next point.
  If no better solutions are found, the step factor is multiplied by the step factor decay (and rounded to an integer value).
- If no better solutions are found, and the step factor is ONE, the optimization is finished
- Variations of the parameterized structure are calculated in various subdirs e.g. v-opt-00001-dx=4500-w=2800

USAGE

Parameter Sweep:

Right click on “Parameter Sweep Setup” – Create Parameter Sweep
Click on Sweep #number
Check Config and Variable Setup
Click “Start”



29.2.17. Parameter Sweep

User Guides
EMPIRE Applications
Releases

Parametric objects and values

OBJECTIVES

- Define parametric objects and values for automatic variations and optimizations.
- Simplify structure definition for unfixed geometries.

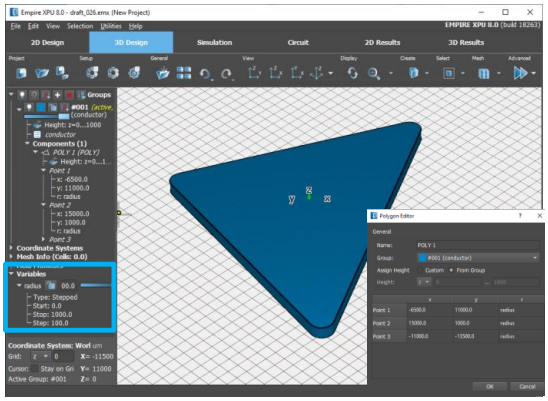
REQUIREMENTS

- Variables are character strings without spaces or special characters, e.g. xvar, y1, ...
- Variables are entered in coordinates or menu entries.
- Mathematical function names should be avoided, e.g. max
- Equations are supported
- Short names are recommended (to be used in simulation folder names)

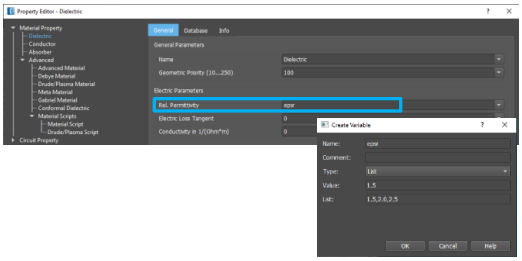
PROCEDURE

- Right click on “Variables” – Create New Variable
- Enter Name, Type, Value and Range
- Enter Variable in component or menu
- Check Range with slider

Polygon radius as parameter



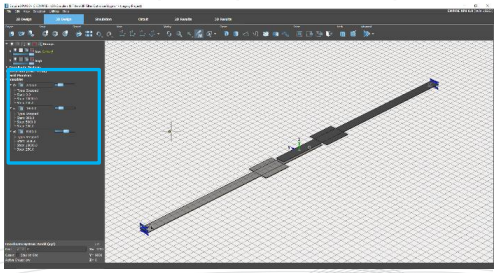
Dielectric as list variable



PARAMETER LIST

- If a parameter has been defined the list is immediately updated and the parameter range can be adjusted.
- The parameter range is asked directly after the definition.
- The parameters can be swept with the slider and the object variation is visualized immediately.
- If the parameters shall be used for optimization they have to be unlocked.
- The **Save Settings** command will store the current parameter settings.
- The **Load Settings** command can be used to set the current parameters to, e.g. optimized values.
- **Remove** will disconnect all variables from the objects or menus and set to current value.

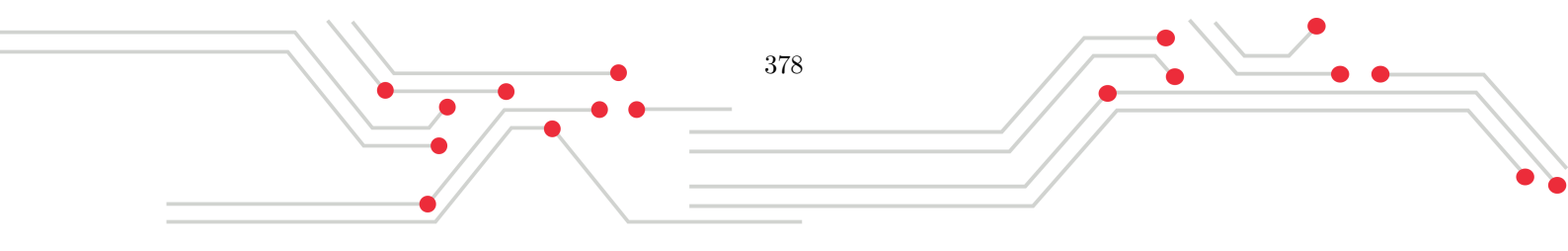
BP filter example with parameters for optimization



PARAMETER AND EQUATIONS

Mathematic functions and definitions can be combined with parameter definitions, e.g.:

- 'asin', 'cos', 'log', 'atan', 'acos', 'sqrt', 'cosh', 'abs', 'sinh', 'pow', 'tan', 'pi', 'log10', 'sin'



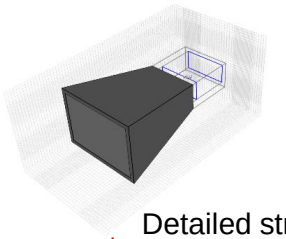
29.2.18. Protected Part

User Guides
EMPIRE Applications
Releases

Read-Protected Part

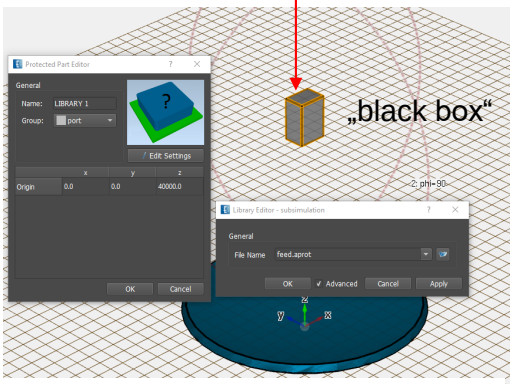
OBJECTIVES

- Create read-protected structure from input files
- Import partial structures as black box for simulation, including e.g. objects, properties, mesh, ports
- Share EMPIRE input files, e.g. for excitation without revealing geometry



Detailed structrure
.aprot
.akey

PROCEDURE

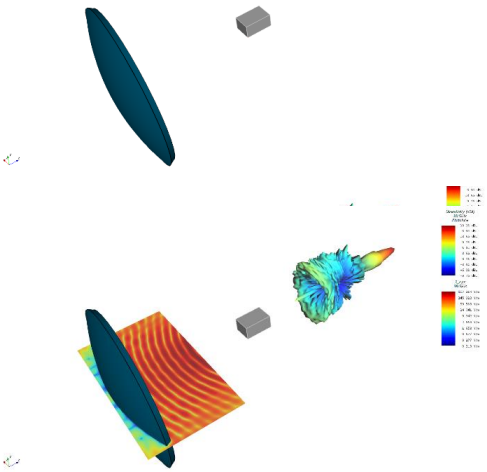


1. Export Protected Part
- Press File – Export – Empire – Read Protected Simulation File (.aprot)
- Optionally, create encryption key (.akey)
- Share .aprot and .akey files

2. Import Protected Part
- Input file < xx>.emx and project folder <xx> available
- Place .aprot and .akey in project folder
- Create Library Element – Substructure – Protected Part
- Specify .aprot file in Element Parameters

Notes

- The imported structure is hidden and represented by a bounding box
- Meshing can not be changed in the volume of the bounding box
- No objects or field monitors can be added inside bounding box volume
- Additional objects (e.g. reflector, scatterer) can be placed outside the bounding box
- Simulation Settings (e.g. frequency range, structure type, boundary conditions) are used from the current project file <xx>.emx



EMPIRE XPU ™ is a product of IMST GmbH, Carl-Friedrich-Gauss-Str. 2, 47475 Kamp-Lintfort, Germany
P +49 2842 981-400, F +49 2842 981-499, E empire@imst.de, I www.empire.de

29.2.19. Remote Processing

User Guides
EMPIRE Applications
Releases

Remote Control

OBJECTIVES

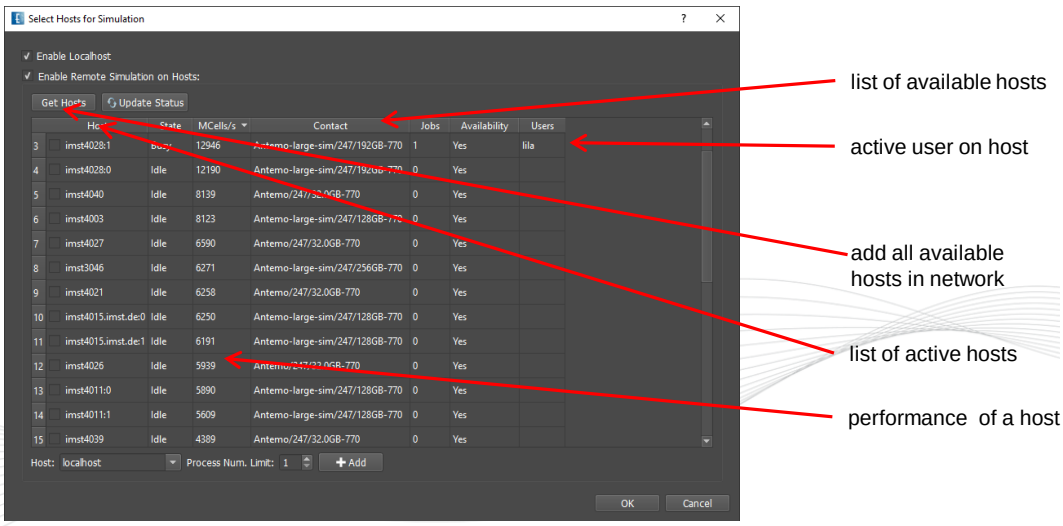
- Distribute different simulation jobs to servers within a network
- Parallel Computing of
  - Different Excitations (sub-1,sub-2 etc.)
  - Different Parameter sets during Optimization
  - Different Structures in Batch Processing

GENERAL REQUIREMENTS

- EMPIRE Service to be started on the remote machine(s) (Windows or Linux)
  - using commands EMPIRE-SERVER (reconectable) or EMPIRE-SERVER-LEGACY
  - from the Utilities - Server – Start (reconnectable) or Server (Legacy) – Start
- empire-rc license needed on the local machine, empire-fs (+ optional nf2ff,ve,wg) license needed on the remote machine
- Communication over TCP/IP and UDP port 42000
- All necessary files are copied from/to the host during simulation
- No Special User Accounts and/or Network Drives needed on the remote machine

USAGE

- Press the **Get Hosts** button to find hosts in the network running the EMPIRE Server
- Enable the hosts you want to calculate on (check the box next to the entity)
- Start Simulation (or Parameter Sweep, Optimization)
- The hosts table will keep you informed about the host status
- The fastest available host will be taken first



EMPIRE XPU ™ is a product of IMST GmbH, Carl-Friedrich-Gauss-Str. 2, 47475 Kamp-Lintfort, Germany
P +49 2842 981-400, F +49 2842 981-499, E empire@imst.de, I www.empire.de

29.2.20. Empire Server v2

User Guides
EMPIRE Applications
Releases

Empire Server v2 (only available with v9.0.x or later)

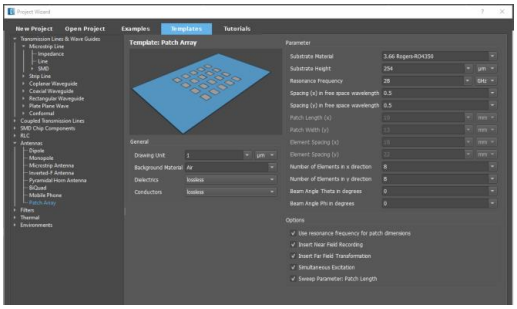
Introduction

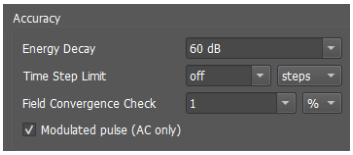
- With the release of Empire 9.0.x, a new server protocol was introduced that changes the way in which personal (office) computers communicate with the servers on the computer cluster
- Until recently, during the complete simulation of an *.emx or *.gym model on the cluster, both the local machine and Empire should be kept running to obtain results; otherwise, the results were lost
- With the new server protocol, this is no longer the case; the software can be terminated locally as soon as the simulation is uploaded and started
- The local version of Empire may then reconnect to the server at a later time and retrieve the simulation results; thus, the new protocol allows for asynchronous communication between clients and servers
- The capabilities of the new server protocol will be explained by way of the example below

Using the New Empire Servers

Setting up an example model

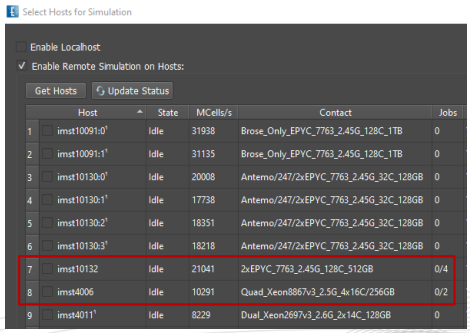
- Start a fresh instance of Empire XPU 9.0.x
- Go to Templates, select Antennas > Patch Array
- Confirm selection with OK
- Save the model on the local PC
- Under the tab “2D Design” go to Simulation Settings > EM Setup
- Set Energy Decay = 60 dB and activate the box “Modulated Pulse (AC only)”; exit with OK[1]



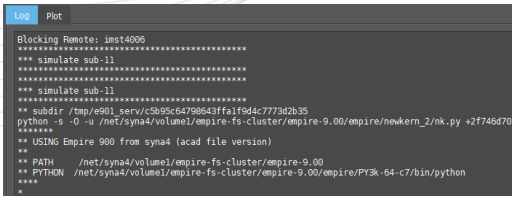


Connecting to the right Servers & Simulating

- Under the tab “Simulation” click on “Host Setup” and uncheck the box “Enable Localhost”
- In the list of hosts available on the cluster, locate the ones that *do not carry* the exponent “1” at the end of the “Host Name”
- These are the servers that support the new communication protocol; as of this writing, the available servers are imst10132 and 4006
- All other servers are now considered as “legacy” ones, that is, they can be used with XPU 9.0.x, but they do not implement the features described herein
- After selecting one of the designated servers, exit with OK, click on “Simulation” on the far-left list and then click on “Start Complete Simulation”
- Confirm all messages with OK
- As soon as the server is blocked and the simulation begins, go to File > Exit Empire
- Wait for 5 minutes so that the simulation may finish



The new servers display in real time the number of simulation jobs currently running on them



[1] These changes in simulation settings are only meant to prolong the simulation time; they are *not* related to the new server protocol

29.2.21. TDR Simulation

User Guides
EMPIRE Applications
Releases

Time Domain Reflectometry

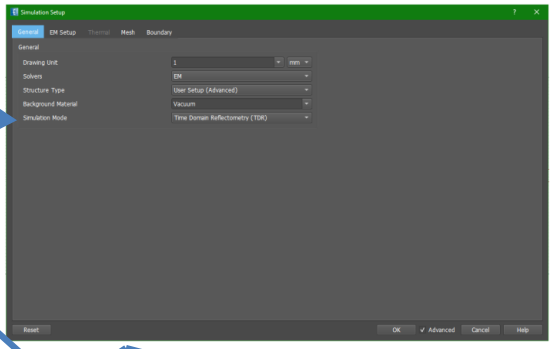
OBJECTIVES

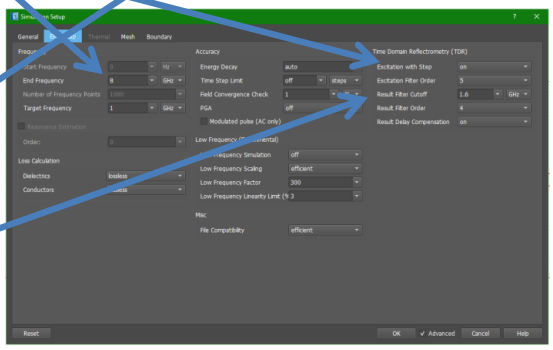
- Time domain reflectometry (TDR) is a diagnostics tool that works like a “waveguided radar”
- A broadband signal is launched by a Port into a system of waveguiding structures [planar transmission lines (TLs), coaxial cables, transitions, etc.]
- Impedance changes are remotely tracked in the spatiotemporal domain via reflected waves
- TDR can be used to verify impedance calculations, detect abrupt impedance changes (faults & defects), and optimize TL discontinuities and transitions between different types of TL

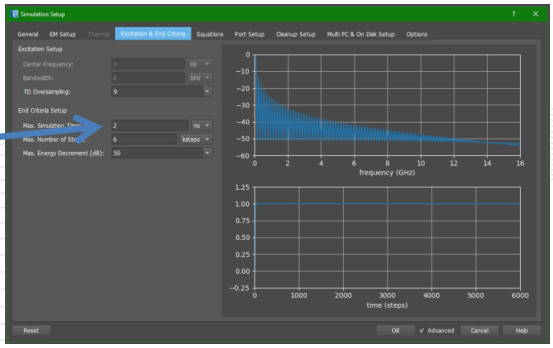
USAGE

Simulation Setup

- Under “General”, select TDR as the “Simulation Mode”
- Let us assume that the microwave circuit operates at some desired frequency, f_X
- The TDR signal needs to be sufficiently wideband in order to provide sufficient resolution in the spatiotemporal domain
- Under “EM Setup”, the “End Frequency” should be set to 8–12 times f_X (3–3.6 octaves)
- Energy decay is irrelevant in TDR simulations; the simulation cannot be terminated on the basis of an energy criterion, since the structure is excited by a step-function and not by a bell-shaped Gaussian pulse
- The “Excitation Filter Order” may remain at the default value of 5
- After the simulation ends, the TDR response is low-pass filtered in order to remove the unnecessary spectral content
- The cut-off frequency of the LPF, f_{CO}, should be set to 1.5–2 times f_X; for example, good results can be obtained with $f_{CO}/f_X = 1.6$
- “Result Filter Order” can be set to either 3, or 4
- “Result Delay Compensation” is set by default to “on” in order to undo the time shift caused by the filter
- Once the model is pre-processed, the user can access “Simulation Setup” under the tab “Simulation”. There, under “Excitation & End Criteria” the user needs to set an appropriate “Max. Simulation Time” and the corresponding “Max. Number of Steps”. The simulation time is the termination criterion and must be set according to the dimensions and material properties of the model. For example, if all dielectric permittivities are unity, then it takes 1 ns for the waves to traverse 300 mm of physical length.







29.2.22. Thermal Simulation

EMPIRE User Guides
Applications
Releases

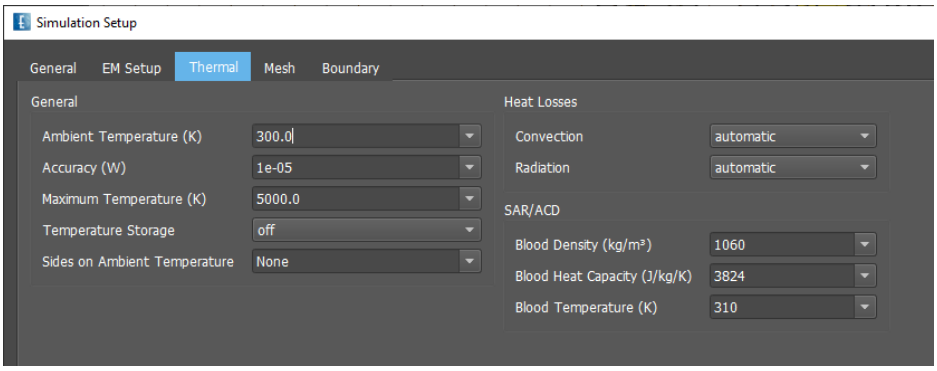
Thermal Solver

OBJECTIVES

- Thermal simulation of RF circuits, including
  - Thermal conductivity of different materials
  - Thermal sources, e.g. transistors, RF losses
  - Heatsinks
  - Convection and radiation cooling
- Human body thermal effects of electromagnetic radiation, including
  - Thermal conductivity of tissues
  - Blood perfusion rate

GENERAL REQUIREMENTS

- EMPIRE XPU™ Software
  - EMP_BP Basic Package license
  - EMP_TS Thermal Solver license



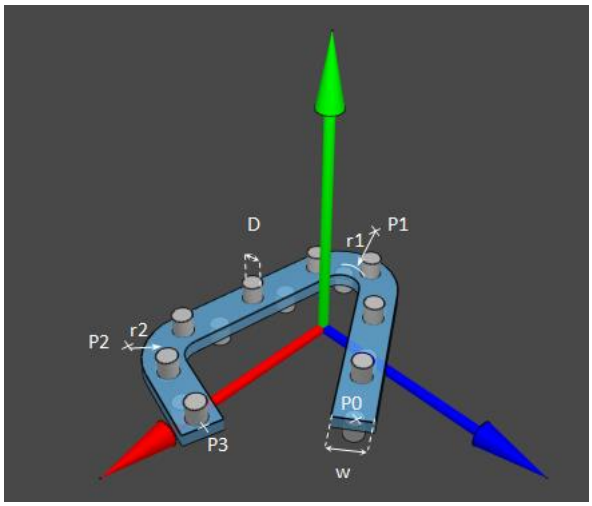
USAGE

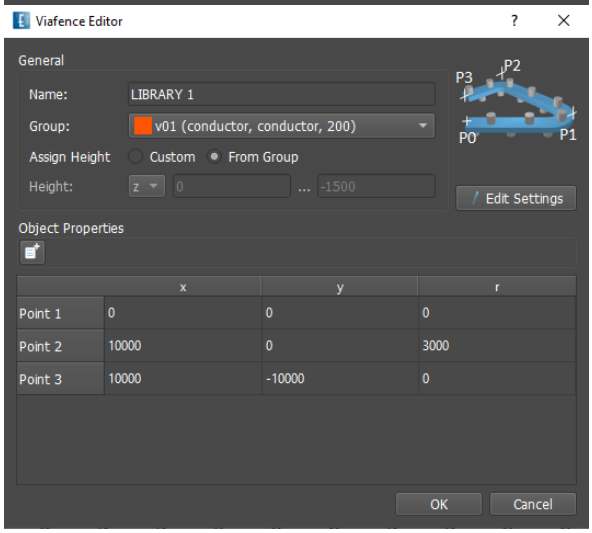
- Select combined “EM+Thermal” or “Thermal” as “Solver” in “General” tab of Simulation Setup
- Enter structure and define thermal parameters for conductors and dielectrics
  - Thermal Conductivity
  - Surface Heatsink and Radiation Coefficient (if applicable)
  - Capacity and Density (for transient solution)
- Define thermal elements
  - heat sinks and sources
  - convection and radiation areas (if applicable)
- Define thermal field display
- For “EM+Thermal” simulation define EM Field Monitors as sources
- Adjust settings for thermal simulation (ambient temperature, accuracy, etc.) in “Thermal” tab
- Create grid and set boundary conditions
- Start the simulation

29.2.23. Via Fence

EMPIRE User Guides
Applications
Releases

Via Fence creation





OBJECTIVES

- Create a shielding fence on a multi-layer PCB
- Automatically arrange vias along a polygon
- Optimize via distribution

USAGE

- Set up layer stack:
  - Group and height for first via layer (active, required)
  - Group and height for second via layer (option)
  - Group and height for catch pad layer (option)

- Create Viafence on first via layer:
  - Create Library object – Layout - Viafence
  - Enter set of polygon corners (points)
  - Optionally, set corner radii in Viafence Editor
  - Click Edit Settings
  - Enter values for Geometry and Groups
  - OK

Remarks

- Viafence object is created on “Via 1 Group” (“None” should be avoided)
- For alternating via diameters D1 and D2 may be different
- For alternating via groups “Staggered Vias” can be set to “on”
- If “Via 2 Group” is None, D1 is used for odd ly and D2 is used for evenly numbered vias
- “Via Distance” (=minimum via spacing) has to be larger than max(D1, D2)
- “Catch Pad Width” has to larger than max(D1, D2)
- If “Fixed via position” is off, the first via position is optimized to maximize density
- If “Catch Pad Group” is set first and last via use a minimum offset of 0.5*w
- If “Catch Pad Group” is “None” first via starts at Point 1 if “Fixed via position” is set

30. Circuit Simulation

EMPIRE XPU™ features a circuit simulator which allows a co-simulation with the EM-FDTD simulation. This is done as part of the post processing where the circuit elements can be connected to EM simulation ports. The results (S-Parameters, Far fields) can be plotted inside the schematic window.

Hint: Circuit Simulation

EM and Circuit Co-Simulation requires Simulation Mode "Sequential Excitation & Y-Parameter-Extraction" to be set in **Simulation Setup - Simulation Mode**

30.1. Circuit Simulation

A pure circuit simulation is possible by placing circuit elements in the schematics window and start a circuit simulation.

- Switch to **Circuit** tab
- Click **Add Schematic**
- Create Circuit elements
- Connect each pin of circuit elements with
  - Wire Connections
  - Terminations, e.g. ground
  - Ports
- Click **Run Circuit Simulation**
- Click **Add Result** place plot and select S-Parameter

See Figure 30.1 for an example.

30.2. EM - Circuit Co-Simulation

To be used after EM-simulation with n excited ports with **Simulation Mode** "Sequential Excitation & Y-Parameter-Extraction" to be defined in the **Simulation Setup**

- Switch to **Circuit** tab
- Click **Add Schematic**
- Click **Empire Ports**
- Create Circuit elements
- Connect each pin of circuit elements with
  - Wire Connections

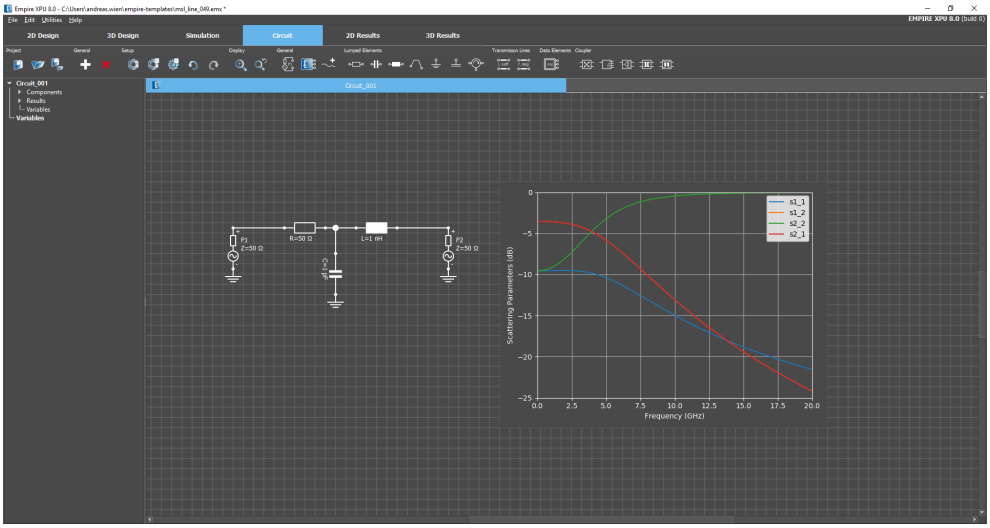


Figure 30.1.: Circuit Simulation Example

  - terminations, e.g. ground
  - Ports

- Click **Run Circuit Simulation**
- Click **Add Result** place plot and select S-Parameter or Far Field

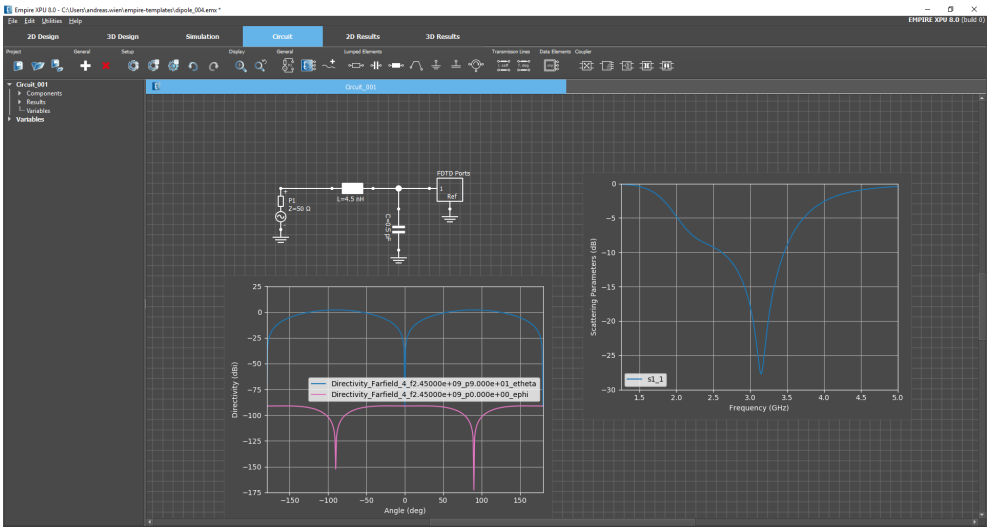


Figure 30.2.: EM-Circuit Co-Simulation Example

See Figure 30.2 for an example (Dipole with matching network)

30.3. EM - Circuit Co-Simulation with Far field superposition

To be used after EM-simulation with

- n excited ports
- **Simulation Mode** "Sequential Excitation & Y-Parameter-Extraction" to be defined in the **Simulation Setup**
- far field monitor with cut plane and component

Usage

- Switch to **Circuit** tab
- Click **Add Schematic**
- Click **Empire Ports**
- Create Circuit elements
- Connect each pin of circuit elements with
  - Wire Connections
  - terminations, e.g. ground
  - Ports
- Click **Circuit Simulation Configuration**
- Select **Enable Far Field Superposition** 30.3
  - **Off** disable far field superposition
  - **Farfield Superposition (Fast)** use far field results of single excitations
  - **NF2FF Superposition (Accurate)** use near field results of single excitations
- Click **Run Circuit Simulation**
- Click **Add Result** place plot and set Plot Type **Far Field**
- Select far field cut plane and component

See Figure 30.4 for an example (Patch Array with divider network)

30.4. Circuit Library

- If circuit elements are created they will be displayed with a symbol in the schematic window
- Each element has one or more pins to be connected with **Wire connections**
- Each element can be picked and placed at arbitrary poistions. To finish the placement press **Esc** key.
- Each element can be edited by double-clicking
- Right click on element to edit or remove this item

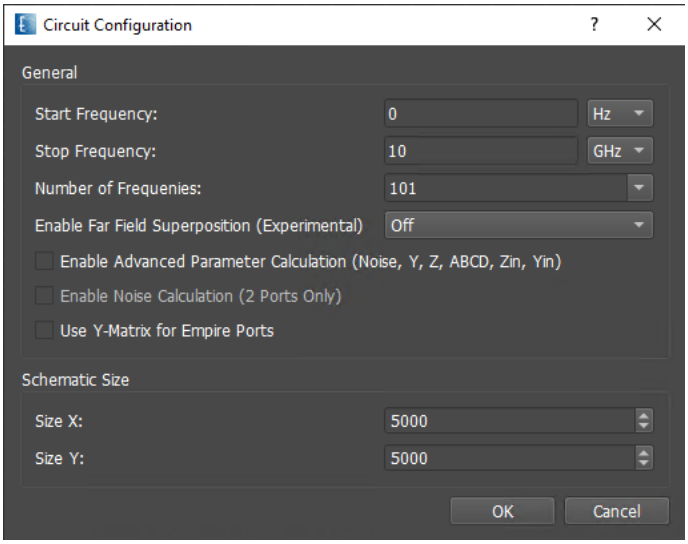


Figure 30.3.: EM-Circuit Configuration

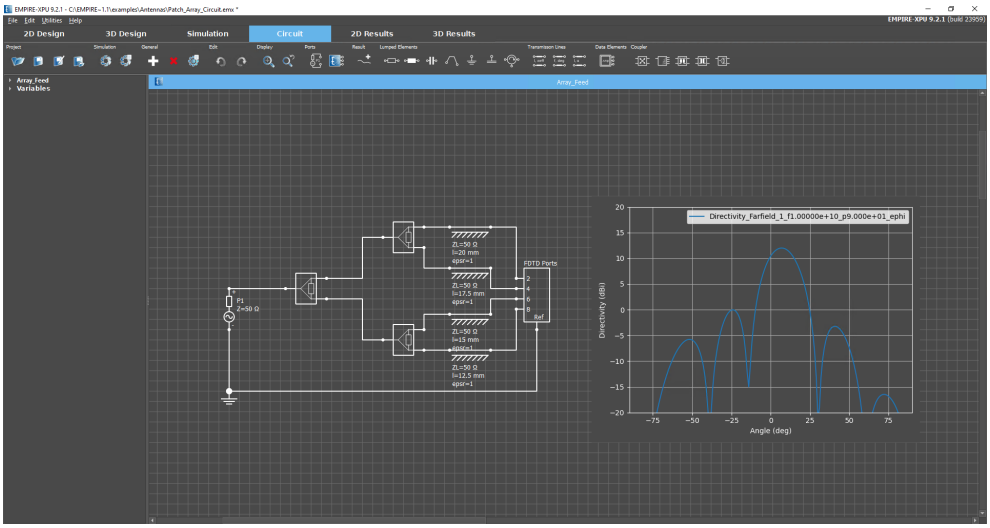


Figure 30.4.: EM-Circuit Co-Simulation with Far Field



Figure 30.5.: Circuit Element Library

30.4.1. General

- **Port** Circuit port to be connected to ciruit elements. Port number, impedance and temperatur can be set.
- **Empire Port** EM simulation port(s). Number is equal to number of excited ports used in EM simulation
- **Add Result** Plot of S-Parameters or far field cut to be placed in schematic window

30.4.2. Lumped Elements

- **Resistor** Ideal resistor in Ohm
- **Capacitor** Ideal capacitor in Farad
- **Inductor** Ideal inductor in Henry
- **Wire Connections** Ideal wire (no delay, no loss)
- **Ground** Short termination
- **Open** Open termination
- **Circulator** Internal impedance can be set for each port

30.4.3. Transmission Lines

Ideal transmission line with line impedance and loss factor to be set in the Parameters

- **Transmission Line (l, ZL, EPSEFF)** Phase shift is determined by length and effective permittivity. Impedance and loss factor may be set.
- **Transmission Line (ZL, DEG, FREF)** Phase shift is determined by angle at specified frequency. Impedance and loss factor may be set.

30.4.4. Data Elements

S-Parameter files (Touchstone format)

- **snp File** n port S-Matrix (n pins). Touchstone file (.snp) has to be located in project folder.

30.4.5. Coupler

Ideal couplers

- **Phase shift coupler** Impedance, coupling factor and phase shift can be specified
- **4x90 degree coupler** Impedance can be set.
- **Wilkinson Divider** Impednace and number of stages can be set.
- **Transformer** Turn ratio can be set.
- **Coupled Inductors** Input and output inductances and coupling factor can be set.

30.5. Circuit Simulation Setup

30.5.1. Circuit Simulation Configuration

Click **Circuit Simulation Configuration** to adjust frequency range and size used for circuit simulation.

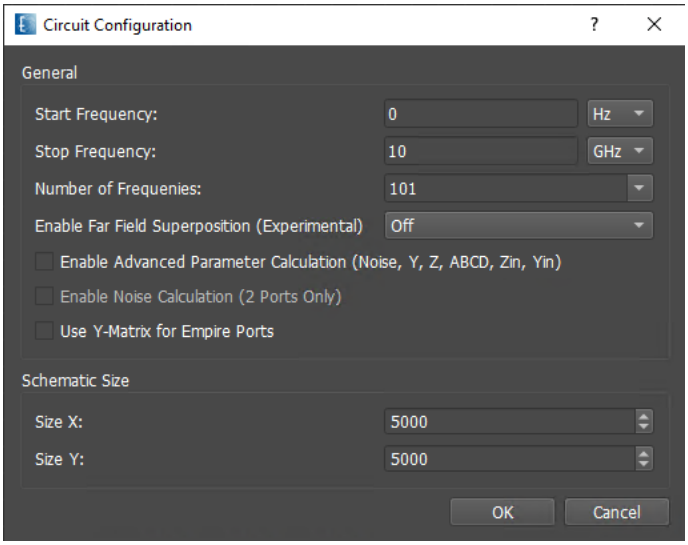


Figure 30.6.: Circuit Simulation Configuration

30.5.2. Circuit Editor Options

Click **Circuit Editor Options** to adjust colors and background for circuit simulation.

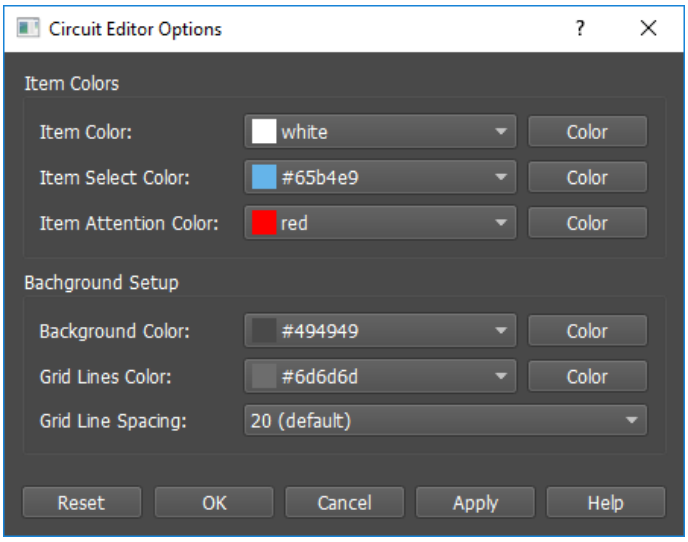


Figure 30.7.: Circuit Editor Options

31. Thermal Simulation

The temperature distribution can be obtained with a thermal simulation based on thermal sources, thermal parameters of objects, heat sinks and thermal simulation parameters. There are two options which can be set in the **Solvers** menu of the **Simulation Setup**

- **THERMAL** - This option runs the thermal solver only. Therefore the thermal power sources have to be known which is usually the case for amplifier, diodes, resistors, etc.
- **EM+THERMAL** - This option runs an EM simulation first and calculates the EM loss at distinct frequencies in volumes. Those losses will be used as the thermal source in a subsequent thermal simulation.

The simulation is carried out for thermal equilibrium by solving the static thermal equations.

31.1. Thermal Simulation Setup

The workflow to setup a thermal simulation is as follows

- Define thermal sources as described in section 20.3
- For each object, check the thermal parameters in the group property settings, see section 18.2.1 or 18.2.2
- If applicable define additional heat sinks (cooling devices), see section 18.4
- Check the global thermal parameters in the **Simulation Setup** - **Thermal** tab, see section 23.3
- Create a suitable mesh for simulation
- Create a thermal field monitor to display the results after the simulation, see setion 19.5
- Start the thermal simulation

Remarks

- **Sources** - It is also possible to use objects with constant temperature as a source. This is defined by an object on a group with **Constant Temperature** property
- **Heat Sink** - If no heat sink is defined, the only cooling mechanism is radiation or convection and can lead to high temperature
- **Ambient Boundary** - It is possible to assign boundaries of the structure bounding box to the ambient temperature which acts as a heat sink with zero thermal resistance.
- **Mesh** - The mesh cells for the thermal simulation is the same as for the EM simulation which is necessary for the combined EM+THERMAL simulation. In case of THERMAL simulation only, the mesh is still derived from the entered frequency range. In this case the end frequency should be set to a low value if the mesh is too dense.
- **Convection and Radiation**
  - are applied at the interface of object surfaces to air which has ambient temperature
  - can be applied as heat sinks, only (not heat sources)

- **Boundary conditions** which are defined in the simulation setup at outermost mesh lines (boundary walls) are not used
- **Sides on constant temperature** apply to outermost object boundaries - not boundary walls
- **Background Material** does not have thermal conductivity, only ambient temperature. Thus a thermal coupling between two distnat objects has to be modelled with a separate thermally conducting material.

31.2. Thermal Solver

EMPIRE XPU™ assumes the surrounding air to have ambient temperature. Thermal sources cause a local temperature rise and a temperature flow to heat sinks is caused by the thermal conductivity of the objects. Radiation and convection from the surface and blood perfusion act as additional cooling mechanisms.

Hint: Thermal Simulation

Thermal simulations in EMPIRE XPU™ are static. A transient solution is not available

The thermal simulation is done by discretizing the static temperature equation on the mesh using the Finite Difference method. This leads to a system of equations with a sparse matrix which is then solved by a Successive Overrelaxation (SOR) method. The progress of the simulation is printed in the log window.

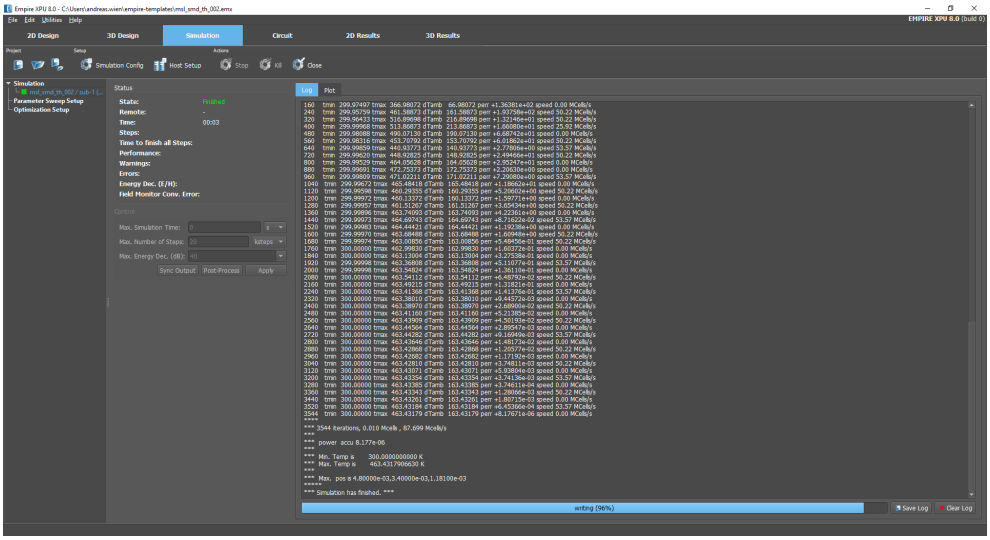


Figure 31.1.: Snapshot of a thermal simulation

The log window shows some thermal parameter statistics and the problem size. Then the solving of the temperature starts by displaying information on the iterations along with current minimum and maximum temperature, current error and speed in cells per second.

Hint: Thermal Simulation End Criterion

The simulation is finished if the error limit which is set in the simulation setup is reached.

The error unit is Watt and is a measure for the accuracy obtained after the iterations. It has to be much smaller than the source power, e.g. 0.001%. In case of a combined EM+THERMAL simulation the source power is printed in the log file, too, before the thermal simulation starts.

32. Body Model Simulation

Body models are usually defined by a volume with an equidistant mesh. To each mesh cell a tissue can be assigned with electrical parameters (Gabriel model) and thermal parameters. In EMPIRE XPU™ two formats are supported which can be loaded into EMPIRE XPU™ or can be visualized and edited with the **Voxel Editor**.

- **Voxel Files** - Empire's native voxel format, file ending **.vxx**. Some models can be found in the **Examples** tree.
- **Raw Files** - Public format, file ending **.raw**. It can be imported into EMPIRE XPU™ as a Zip archive (file ending **.zip**) if it contains the **.raw**-file and an associated tissue file **.txt**

The workflow for body modeling can be subdivided in to the following steps:

- In EMPIRE XPU™ load one of the voxel file models defined above, see section 32.1
- In EMPIRE XPU™ define an excitation scenario (e.g. antenna), simulation parameters and a suitable meshing
- Optionally for thermal solution, define a thermal source from EM loss which covers the voxel model, see section 20.3.1
- Define a field monitor (EM full volume or EM field volume) for ACD/SAR/EIAV calculation and/or thermal display, see section 19.1
- Run the EM simulation or a combined EM+Thermal simulation
- Adjust the field monitor display parameters for the desired averaging method for ACD/SAR/EIAV calculation
- Display the results in the **3D Results** tab or see distributions in **2D Results**

32.1. Load Voxel File

The loading of a voxel file will be explained by an example file (monkey model) which is available in the example tree: `C:\EMPIRE_XPU_9.x.x\SAR & ACD & EIAV\Monkey Exposed to Plane Wave`

This example folder contains a body model `monkey.vxx` which consists of different tissues resolved into an equidistant mesh of 1mm spacing. The overall size is 230mm x 161mm x 717mm. In order to follow the steps the voxel file should be copied into a fresh directory.

Step 1:

1. Open the Windows Explorer and create a new folder **monkey**
2. Copy the file `monkey.vxx` into the new folder
3. Start EMPIRE XPU™ from the desktop, set unit to mm and press **New Project**
4. Save the project into the new folder **monkey**

32.1.1. Import Voxel File

The voxel file is inserted into EMPIRE XPU™ by

- Creating the library element **Substructure - Voxel Model**
- Defining an anchor **Origin**
- Setting parameters with **Edit Settings**

For compatibility reasons voxel files can also be defined by creating a group with property **Advanced Property - Voxel Model** and a box which defines an anchor point, defined by the minimum coordinates x0, y0, z0.

The parameters of the voxel model are explained in the chapter **Object properties** 18.6.1.

Step 2:

1. Click **Create Library Element**
2. Select **Substructure - Voxel Model**
3. Set at origin (x0=0, y0=0, z0=0)

Hint

The Origin of the library element (x0,y0,z0) defines the insertion point of the voxel file.

In the Draft mode the voxel file is displayed by slices defined in the voxel property definition. In the 3D mode the voxel file can be displayed with as a surface, slices or selected tissues.

In Figure 32.1 the monkey model is displayed.

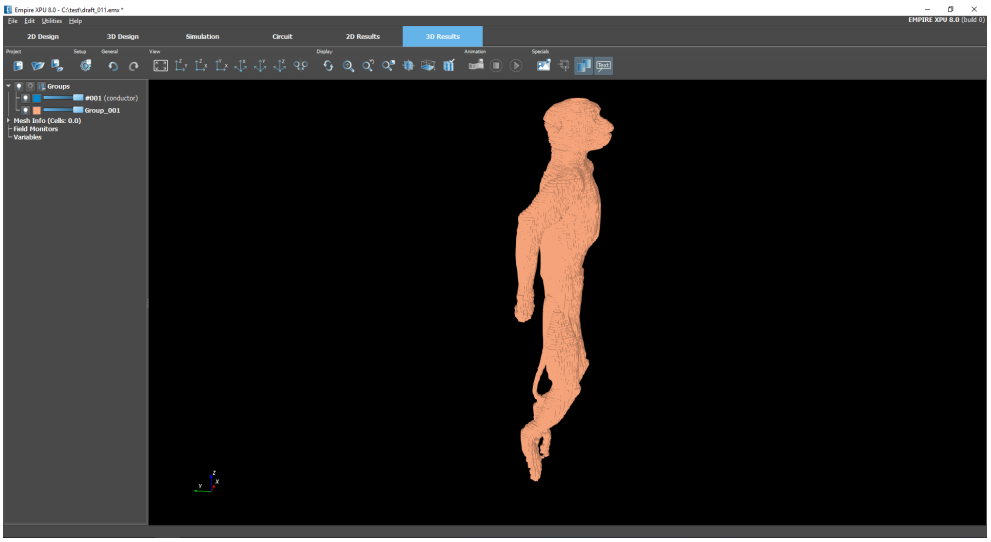


Figure 32.1.: View of the monkey model

Part IV.

Command and Setup Reference

33. Actions

33.1. Boolean

---

1  Merge

Operates on Geometric Objects, Library Object Objects.

```
An object is created which is defined by the total volume
of the selected OBJECT(s)

Note: If the objects' type are not equal they will be converted into
Polygons or Solids
```

---

2  Subtract first-second

Operates on Geometric Objects, Library Object Objects.

```
The second selected OBJECT will be subtracted from the
first selected OBJECT

Note:
If the objects' type are not equal they will be converted into
Polygons or Solids
```

---

3  Multiple Subtract

Operates on Geometric Objects.

```
Selected small OBJECT(s) will be subtracted from selected large OBJECT(s)

The large objects must entirely cover the small ojects.

Note:
If the objects' type are not equal they will be converted into
Polygons or Solids
```

---

4  Intersect (Ctrl y)

Operates on Geometric Objects.
Shortcut: <Control-y>

```
An object will be created which is defined by the common
volume of the selected OBJECT(s).

Note: If the objects' type are not equal they will be converted into
      Polygons or Solids
```

33.2. Clear

---

5  Escape (Esc)
Shortcut: Esc

```
All selections (objects or operators) will be de-selected.

Pressing this button clears all stack entries.

Equivalent to Esc key.
```

33.3. Create

---

6  Create Box

Operates on Arrow Objects.

```
Creates a BOX from an ARROW.

Forward usage:
- Press Button
- Enter BOX co-ordinates on the left / Enter Arrow
- Press OK

Backward usage:
- Enter an arrow in drawing area (drag left mouse button)
- Press Icon

Note:
The group height has to be normal to the current view.

Note:
Handles represent edges of the box and can be used to
stretch, copy or move with the left, middle or right mouse
button if active (green).
```

---

7  Create Cylinder (Circular)

Operates on Arrow Objects.

```
Creates a 3D circular cylinder defined by an arrow.

The starting point of the ARROW is the origin of the circle
and the length defines its radius.

Forward usage:
- Press Button
- Enter co-ordinates on the left / Enter an arrow
- Press OK

Backward usage:
- Enter an arrow in drawing area (drag left mouse button)
- Press Icon

Note: The group height must be normal to the current view.

Note:
Handles represent midpoint of the circle and can be used to
stretch, copy or move with the left, middle or right mouse
button if active (green).

Note:
Created Cylinders are listed in "Components" in Side Menu
```

---

8  Create Polygon

Operates on Point Objects.

```
Creates a closed POLYgon from a set of POINTS.

Forward usage:
- Press Button
- Enter Point co-ordinates on the left
- optionally, insert points by pressing up/down icons

Backward usage:
- Enter set of points in drawing area (left mouse button)
- Press Icon

Note: The group height has to be normal to the current view

Note: Each corner can be rounded by
- Open the left list + Structure + Poly
- Select desired Poly
- Enter radius value of each corner

Note:
Handles represent corners of the polygon and can be used to
stretch, copy or move with the left, middle or right mouse
button if active (green).

Note:
Created Polygons are listed in "Components" in Side Menu
```

9  Create Library Object

```
Starts the Library wizard for object definition

Inserts an Library Object into the structure which
position is defined by one or more points, group height
and more parameters to be entered in the object menu.

Forward usage:
- Press this button or open the Object Library tree
  and select one of the predefined shapes
- Enter the Point(s) and Parameters

Backward usage:
- Enter Point(s) (click left mouse button in workspace)
- Press this button and select predefined shape
- Enter Parameters


Note:
Values may be entered parametrically, e.g. for automatic
variation

Note:
Handles represent points P0, P1, ... of the object and can be
used to stretch, copy or move with the left, middle or right
mouse button if active (green).

Note:
Created Objects are listed in "Components" in Side Menu
```

10  Create Source

```
Starts the Library wizard for source definition

Sources are:
- Ports (lumped and feedline types)
- Plane wave
- Thermal sources

Note:
Position and orientation can be adjusted with the values
displayed in the side menu
Note:
Further parameters can be set in the Parameters menu

Ports:
Inserts a source into the structure which
- perpendicular and propagating direction can be chosen by switches
- Start Point, length and height can be entered with coordinates
- height or perpendicular position may be defined by the
  current group height

Exception:
Perpendicular Port
- has the direction of the current group arrow
- cross section is defined by Point 0 and Point 1

Note:
Planar Ports (MSL, CPW, ...) Ports get the properties of the
current group. Coaxial, Waveguide and Lumped Ports
have their own properties.

Note:
Handles represent P0 and P1 of the sources and can be used to
stretch, copy or move with the left, middle or right mouse
button if active (green).

Note:
Created Sources are listed in "Sources" in Side Menu
```

---

11  Create Circuit Element

```
Starts the Circuit Element wizard

Usage:
- Add schematic circuit elements to the model
- Use Wires to connect circuit elements to excited ports

Method:
- The scattering matrix of the model is calculated by FDTD without circuit elements
- The circuit elements are added in frequency domain after postprocessing

Note:
Created Elements are listed in "Circuit Elements" in Side Menu
```

---

12  Create Field Monitor

```
Starts the Library wizard for Field Monitor definition

Defines a Field Monitor with size and position defined by
one or more points which are displayed in the side menu.
More parameters to be entered in the object menu.

Usage:
- Press this button or open Field Monitor - Create in the Side Menu
- Select one of the available Field Monitors
- Enter the Point(s) and Parameters

Note:
- Storage parameters have to be defined before simulation
- Calculation parameters have to be defined before postprocessing
- Display parameters can be adjusted after postprocessing

Note:
Created monitors are listed in "Field Monitors" in Side Menu
```

33.4. Delete

---

13  Delete Mesh

Operates on Discretization Objects.

```
Manual Meshing:
Deletes all lines (fixed and non-fixed) on the selected GRID bar.

Note:
To select press left mouse button on GRID bar.
To de-select a GRID bar, press ESC.
Undo will work.
```

---

14  Delete Mesh Filling

Operates on Discretization Objects.

```
Manual Meshing:
Deletes all non-fixed lines (thin red lines) on the selected GRID bar.


Note:
To select press left mouse button on GRID bar.
To de-select a GRID bar, press ESC.
```

33.5. Design

---

15 Simulation Setup

Modify settings, like:

- Unit
- Structure Type (Antenna, Filter, Inductance,...)
- Resolution (Coarse, Fine, ...)
- Energy decay (End criteria)
- PGA (Perfect Geometry Approximation)
- Frequency range used for DFT
- Loss Calculation Model
- Boundary Conditions

More advanced simulation settings are available in the side menu:
Simulation Setup

---

16 Mesh Setup

Open Mesh Options Menu to adjust parameters:

General: - 2D/3D mesh, accuracy, flat metal, slopes, inner structure
Axis Setup - directive settings, global meshint
Simulation Box: fixed user defined limits
Additional Space: enlarge simulation domain

More mesh settings are available in the side menu Automesh Setup

---

17 Simulation Control

This operation starts the Simulation Control with access to
Simulation
Preprocessing
Postprocessing
Batch Processing
Parameter Sweep
Optimization
Tuning

---

18 Store Variant

This operation

- saves the structure to a new variant folder
- moves results to a new variant folder

33.6. Display

---

19 Zoom Extents (z)
Shortcut: z

```
Displays the drawing to the overall extents.

Note:
Zoom may influence current cursor snap.
```

---

20  Restore View

```
Swap between current and last view
```

---

21  Save View

```
Saves current zoom area.
Up to 3 zoom areas can be stored and displayed
by pressing "Restore View"
```

---

22  Zoom Window (z)

Operates on Arrow Objects.
Shortcut: z

```
Zoom into the region which is spanned by the entered ARROW.
```

---

23  Redraw

```
- Updates the drawing with the current database.
- Deletes Snap co-ordinates which have been added
  by selecting an object
```

33.7. Edit

---

24  Editor Options

```
Modify settings, like:

- Background colors for Draft and 3D Mode
- Draft mode cursor appearance
- 3D mode display mode and light position
- Layout and 3D Data Import/Export scale factors
- Default polygon attributes
- Snap and Object selection

More advanced options are available in side menu:
Advanced - Editor Options, e.g.
- Point and Arrow appearance
- Legend and Measurement
- Defaults for Bond wires, Solids, via fences
- Default operations

- Discretization
```

---

25  Delete (Del)

Operates on Geometric Objects.
Shortcut: <Delete>

```
All selected objects will be erased.
Undo will work.
```

---

26  Open Object (o)

Operates on Geometric Objects.
Shortcut: <o>

```
Open selected Object for editing in side menu
```

---

27  Edit Parameters

Operates on Port Objects.

```
Edit parameters of the selected Library Port or Library Object.
```

---

28  Set Port Number(s)

Operates on Port Objects,operand Number Objects.
Shortcut: <equal>

```
Changes the Port number of the selected library port

Forward usage:
- Select Port
- Press Button
- Enter new Port number

Backward usage:
- Type number
- OK (or press right mouse button)
- Select Port
- Press Button
```

---

29  Set Parameter by Equation

Operates on Port Objects,operand Equation property Objects.
Shortcut: <slash>

```
The respective parameter of the selected Library
Element will be set to the entered equation.

Examples:
+ Select Lumped port
+ Press "Modify: Set Parameter by Equation"
+ Enter w=100, OK
+ Press "Esc"

+ Select Sphere
+ Press "Modify: Set Parameter by Equation"
+ Enter d=$a, OK
+ Enter Parameter range, value and step
+ Press "Esc"

Useful to modify multiple library objects at the same time.
```

---

30  Undo
Shortcut: u

```
Cancel last action.

Multiple Undos are possible.
```

---

31  Redo
Shortcut: r

```
Recover a cancelled action.

The possible number of redos is equal the numbers of undos.
```

---

32  Edit Point/Arrow

Operates on Point Objects, Arrow Objects, Number Objects.

```
Edit co-ordinates of entered ARROW/POINT/NUMBER
```

33.8. File

---

33  Edit Substructure

Operates on Substructure Objects.
Shortcut: e

```
Creates a BOX from an ARROW.

Forward usage:
- Press Button
- Enter BOX co-ordinates on the left / Enter Arrow
- Press OK

Backward usage:
- Enter an arrow in drawing area (drag left mouse button)
- Press Icon

Note:
The group height has to be normal to the current view.

Note:
Handles represent edges of the box and can be used to
stretch, copy or move with the left, middle or right mouse
button if active (green).
```

---

34  Save (Ctrl-s)
Shortcut: <Control-s>

```
Saves the current input data into a the standard
input file with ending .gym
```

---

35  Import

```
This operation imports external geometry files.
Main supported formats are:

2D geometry:
- DXF v12 layout
- Gerber 247X layout
- GDS2 layout

3D geometry:
- STL surface triangles
- GYM native format EMPIRE
```

33.9. Group

---

36  Open Current Group

```
Adjust group settings

Open the current group for the purpose of
+ Set / Add / change Properties
+ Set group height
+ Adjust color, style, visibility, ...
+ Create, copy, activate groups
```

33.10. Mesh

---

37  Create Mesh

```
Starts the Automatic Meshing Engine, to e.g. check the structure
after discretization.

Depending on the entered objects, group settings, mesh hints
and global parameters in the Simulation Setup a grid is
generated automatically.

This operation is also executed on pressing "Convert and Save" or
"Start simulation", Parameter Sweep or Optimization unless the
Discretization is set to "Manual".

The Automesh Parameters may be adjusted in the mesh options.
Advanced meshing parameters can be adjusted if the
Discretization is set to "User-defined"
```

---

38  Insert Symmetric

Operates on Discretization Objects,operand Number Objects.
Shortcut: <equal>

```
Inserts Un-fixed lines in the marked interval with a
spacing of the entered NUMBER on the selected GRID bar.

Note:
The spacing is varied to ensure a symmetric filling.

Selection: NUMBER and GRID bar
Undo will work.
```

---

39 Subdivide

Operates on Discretization Objects,operand Number Objects.
Shortcut: <slash>

```
Inserts Un-fixed lines in the marked interval.
The region is subdivided into N equal intervals,
where N is the entered NUMBER.

Selection: NUMBER and GRID bar
Undo will work.
```

---

40 Extract Points

Operates on Discretization Objects,operand Arrow Objects.

```
Points will be extracted from GRID lines
covered by the entered ARROW.

Those points can be used, e.g. to store a
discretization into a variable (Shift + F1)
for a later re-use (F1 to recall).
```

---

41 Meshed 3D Structure

```
The structure is discretized using the current grid and
displayed in the 3D rendered mode. To return to the
drawing mode, press 'Draft' Display.

The original strucutre will not be modified.

Note: Recommended for checking staircase approximation.
```

33.11. Misc

---

42 Change Sign

Operates on Number Objects.
Shortcut: m

```
The entered NUMBER is multiplied by -1.
Selection: NUMBER
```

---

43 Edit Text

Operates on Geometric Objects,operand Text Objects.
Shortcut: <equal>

```
Sets the current TEXTstring to the object
Selection: OBJECT and TEXT
```

---

44 Push Regular Expression

Operates on Text Objects.

```
All objects which properties match the entered TEXT are selected.

Usage:
- Press the t on the keyboard
- Enter text string in the bottom line
- Press button

Note:
Replacers (*, ?, []) may be used.
```

---

45 Reciprocal

Operates on Number Objects.
Shortcut: i

```
The entered NUMBER is replaced by its reciprocal value.
Selection: NUMBER
```

---

46 Toggle Mode

Operates on Measure Objects.
Shortcut: i

```
Changes style of outline measurement.
```

33.12. Mode

---

47 Change To Poly

Operates on Rotpolygon Objects, Bondwire Objects.

```
Converts the selected LINPOLY or ROTPOLY to a (closed) POLY
with the current group height.
```

33.13. Origin

48  Change Origin

Operates on Point Objects.

```
Move origin to the entered point.

The origin is indicated by encircled crosses on bottom
and right grid bars.

Note:
The origin may be reset with Menu View - Reset Origin
```

33.14. Polygons

49  Delete Points

Operates on Polygon Objects.

```
The active (green) handles of the selected POLY will be deleted.

Note:
Handles can be activated (green) / de-activated (red) by clicking
them with left mouse button. Undo will work.
```

50  Insert Point(s)

Operates on Polygon Objects,operand Point Objects.

```
Insert entered POINTS between two active (green) handles.

Selection: POLY and POINTs

Note:
If more than one point should be inserted the sequence of
points have to coincide with the polygon orientation.
```

51  Revolve Polygon around Axis

Operates on Polygon Objects,operand Arrow Objects.

```
Revolves a cross section of a polygon around an axis.

The rotational axis is defined by a 1D ARROW
and the first value of the layer height.

Note:
The entered ARROW must be in the plane of the poly cross section
directed in either x,y, or z direction.
```

33.15. Select

---

52 Select Enclosed (e)

Operates on Arrow Objects.
Shortcut: e

```
Selects all objects which are enclosed by the entered ARROW
and activates all handles of an object.

Note:
Objects on a locked or switched-off group are disregarded.
```

---

53 Select Overlapping (p)

Operates on Arrow Objects.
Shortcut: p

```
Selects all objects which are partially enclosed
inside the region defined by the entered ARROW.

Activates objects' handles which are inside the area
spanned by the arrow.

Note:
Objects on a locked or switched-off group are disregarded.
```

---

54 Last Selection(l)
Shortcut: <l>

```
Select Objects most recently used by eg. move or copy.
```

33.16. Transform

---

55  Move (+)

Operates on Polygon Objects, Solid Objects, Box Objects,operand Arrow Objects.
Shortcut: <plus>

```
Shifts the selected OBJECTs acccording to the entered ARROW.
Selection: OBJECT and ARROW

Forward usage:
+ Press Icon
+ Select object(s)
+ Enter shift arrow

Backward usage:
+ Enter shift arrow
+ Select object(s)
+ Press Icon
```

---

56  Copy (*)

Operates on Polygon Objects, Solid Objects, Box Objects,operand Arrow Objects.
Shortcut: <asterisk>

```
Copy selected OBJECT(s) according to the entered ARROW(s).

Selection: OBJECT and ARROW


Forward usage:
+ Press Icon
+ Select object(s)
+ Enter shift arrow

Backward usage:
+ Enter shift arrow
+ Select object(s)
+ Press Icon
```

---

57  Multiple Copy

Operates on Geometric Objects, Discretization Objects.

```
Copy selected OBJECT(s) according to an array expression:

mcopy(dx=a,dy=b,dz=c,nx=m,ny=l,nz=p)

where a,b,c are the shifts in x,y,z direction
and  m,l,p are the numbers of copies in x,y,z direction

Note:
More complex copy operations are possible using a map function, e.g.

'mcopy(dx=45,nx=8,map= lambda x,y,z: (100*cos(x/180*3.1415),100*sin(x/180*3.1415),0))'

copies an object 8 times around a circle with an angle step of 45 degrees
```

---

58  Mirror (&)

Operates on Polygon Objects, Discretization Objects, Solid Objects, Box Objects,operand Arrow Objects.
Shortcut: <ampersand>

```
The selected OBJECTs/GRID lines are mirrored at a 1D ARROW.
The selected OBJECTs/GRID lines are deleted.

Selection: OBJECT/GRID lines and ARROW

Note: Only 1D arrows (parallel to main axes) are permitted
for mirror operation.
```

---

59  Copy & Mirror (bar)

Operates on Polygon Objects, Discretization Objects, Solid Objects, Box Objects,operand Arrow Objects.
Shortcut: <bar>

```
The selected OBJECTs/GRID lines are mirrored at a 1D ARROW.
The selected OBJECTs/GRID lines are maintained.

Note:
Only 1D arrows (parallel to main exes) are permitted
for mirror operation.

Selection: OBJECT/GRID lines and ARROW
```

---

60  Rotate

Operates on Polygon Objects, Solid Objects, Box Objects, Discretization Objects,operand Point Objects.

```
The selected OBJECTs will be rotated around the
entered POINT which acts as the rotational axis.

The rotation angle is entered positive for counter-
clockwise rotation.

Original objects are deleted.

Note:
Angle may be entered parametric, e.g. a
Multiple angles are possible, e.g. 20@3, generating 3 copies
```

---

61  Copy & Rotate

Operates on Polygon Objects, Solid Objects, Box Objects, Discretization Objects,operand Point Objects.

```
The selected OBJECTs will be rotated around the
entered POINT which acts as the rotational axis.

The rotation angle is entered positive for counter-
clockwise rotation.

Original objects are maintained.

Note:
Angle may be parametric, e.g. a
Multiple angles are possible, e.g. 20@3, generating 3 copies
```

---

62  Stretch (/)

Operates on Polygon Objects, Solid Objects, Box Objects,operand Arrow Objects.
Shortcut: <slash>

```
The green handles of the selected OBJECTs will be stretched
according to the entered ARROW.

Hint: To select/deselect handles click them with the left mouse button.
Hint: Handles can also be selected in the Draft 3D Iso View
Hint: A group of handles can be selected by operation
          "Select overlapping" entering the range arrow from top
           left to right bottom.

Selection: POLYs and ARROW

Forward usage:
+ Press Icon
+ Select object
+ Select corners/edges to be stretched
+ Enter shift arrow

Backward usage:
+ Select object
+ Select corners/edges to be stretched
+ Enter shift arrow
+ Press Icon

Hint: If the shift arrow starts at one active handle
pressing the icon is obsolete.
```

---

63  Set Height(=)

Operates on Polygon Objects, Box Objects,operand Arrow Objects.
Shortcut: <equal>

```
Set new height of the selected POLYs defined by the entered
ARROW.

The ARROW must be a 1D arrow, perpendicular to the polygon
cross section.

Selection: ARROW and POLY(s)
```

33.17. View

---

64  Fullscreen

```
Displays the current drawing in a 3D view.
Lines parallel to the z-axis are displayed as vertical.
```

---

65  Side View

```
Displays the current drawing from the right.
z and y co-ordinates will be displayed.
Normal direction is + x.
```

---

66  Front View

```
Displays the current drawing from the front.
z and x co-ordinates will be displayed.
Normal direction is - y.
```

---

67  Top View

```
Displays the current drawing from the top.
x and y co-ordinates will be displayed.
Normal direction is + z (standard view).
```

---

68  Iso X View

```
Displays the current drawing in a 3D view.
Lines parallel to the x-axis are displayed as vertical.
```

---

69  Iso Y View

```
Displays the current drawing in a 3D view.
Lines parallel to the y-axis are displayed as vertical.
```

---

70  Iso Z View

```
Displays the current drawing in a 3D view.
Lines parallel to the z-axis are displayed as vertical.
```

33.18. Advanced

33.18.1. Advanced/Arrows

---

71  Arrow Concat (+)

Operates on Arrow Objects,operand Arrow Objects.
Shortcut: <plus>

```
Combines the entered ARROWs to a resulting one by vectorial addition.
```

---

72  Arrow Multiple (&)

Operates on Arrow Objects,operand Number Objects.
Shortcut: <ampersand>

```
Generates N arrows with the lengths a, a/2, ..., a/N where
 a is the length of the entered ARROW.
 N is the entered NUMBER

Selection: ARROW + NUMBER
```

---

73  Arrow Scale (*)

Operates on Arrow Objects,operand Number Objects.
Shortcut: <asterisk>

```
Scale the entered ARROW by a factor.
The factor has to be entered as a NUMBER

Selection: ARROW and NUMBER
```

---

74  Arrow Subdivide (/)

Operates on Arrow Objects,operand Number Objects.
Shortcut: <slash>

```
Generates N subsequent arrows with the lengths a/N where
 a is the length of the entered ARROW
 N is the entered NUMBER

Selection: ARROW + NUMBER
```

---

75  Create Arrow (-)

Operates on Point Objects,operand Point Objects.
Shortcut: <minus>

```
Two entered POINTs create an ARROW.
Selection: POINT and POINT
```

---

76  Negate (m)

Operates on Arrow Objects.
Shortcut: m

```
Changes tip with origin of the entered ARROW.
```

---

77  Return Center Point

Operates on Arrow Objects.

```
Extracts the Midpoint of the entered ARROW

Useful, e.g. to define a centered rotational axis
```

33.18.2. Advanced/Boxes

---

78  Copy and Edit Box

Operates on Box Objects,operand Arrow Objects.
Shortcut: <percent>

```
Creates new BOX(es) with the cross section of the entered ARROW(s).

The height is copied from the selected BOX.

Selection: BOX and ARROW(s)
```

---

79  Edit Box

Operates on Box Objects,operand Arrow Objects.
Shortcut: <equal>

```
Modify the cross section of the selected BOX according
to the entered arrow.

If the entered arrow is 1-dimensional the cross section will be
modified in this direction, only.

Selection: BOX and ARROW
```

33.18.3. Advanced/Create

---

80  Create Dimensioning

Operates on Point Objects.

```
Creates a legend with a default text after entering an ARROW.
or
Creates a measurement arrow after entering 3 POINTS.

The first POINT defines the position of the measurement number,
the subsequent POINTS define the measurement distance.

Note:
The first two POINTS should be orthogonal to the measurement line.

Note:
The measurement has no effect on simulation.
```

81  Via Fence

Operates on Polygon Objects.

```
A Via Fence will be generated for the selected Polygon
using the selected Handles to identify parts.

Note: At least 2 handles have to be selected (green squares)

Note: If not already available Pad and Via groups will be created
which height and property have to be set

See Advanced/Editor Options/Via Fence for details.
```

33.18.4. Advanced/Design Check

82  Area Measurement

Operates on Polygon Objects, Box Objects.

```
Calculates the length of the selected POLYGON between two active (green) handles
```

83  Check Distance

Operates on Geometric Objects,operand Number Objects.

```
Checks if distance of two objects' centers are below a
certain limit defined by the entered number.

If the distance is above the limit the objects will remain
selected (highlighted).

The center is defined by the object's bounding box.
```

84 Check Integrity

Operates on Geometric Objects.

```
The selected OBJECT is checked if a single volume will
be obtained after discretization using the current grid.

Note: useful to check if current grid density is sufficient
to resolve thin solid objects without generating gaps.
```

---

85 Check Polygon Point Spacing

Operates on Polygon Objects,operand Number Objects.

```
Selects points of a polygon with distance below the entered number.
```

---

86 Check Symmetry

Operates on Arrow Objects.

```
Selected objects are checked if they are symmetric
to the entered 1D ARROW.

Objects which are not symmetric remain highlighted.

Selection: OBJECTs and 1D ARROW
```

---

87 Create Connectivity Groups

Operates on Geometric Objects.

```
This operation performs a connectivity check of the
selected objects and copies the connected line pathes to new groups

("Signal_n" where n is the number of detected signal path).

If no connection is found the object(s) are copied
to a new group "Unconnected"

Example
To check a multi-grouped printed circuit:
- Select all lines and vias
- Press "Copy to signal group"
- Switch off all groups and switch on one of the
  signal groups to check the detected connection.
```

---

88 Create Connectivity Groups (discretized)

Operates on Geometric Objects, Library Object Objects.

```
This operation performs a connectivity check of the
selected objects and copies the connected line pathes to new groups

In contrast to usual "Copy to Signal groups" all objects
will be first discretized.
```

---

89  Length Measurement

Operates on Polygon Objects, Solid Objects.

```
Calculates the length of the selected POLYGON between two active (green) handles
```

---

90  Map Object on Mesh

Operates on Box Objects, Polygon Objects, Solid Objects, Port Objects.

```
The selected OBJECT(s) are discretized using the current grid.

The original object can be recovered by using UNDO.

Note: Only recommended for checking staircase approximation.
```

---

91  Mesh Info

```
The selected OBJECT is checked if a single volume will
be obtained after discretization using the current grid.

Note: useful to check if current grid density is sufficient
to resolve thin solid objects without generating gaps.
```

---

92  Show Mapped Object

Operates on Geometric Objects.

```
Displays the discretized shape of the selected OBJECT(s)

The selected OBJECT(s) will not be changed.

Use REDRAW to erase discretized view.
```

33.18.5. Advanced/Edit

Advanced/Edit/Convert

---

93  Convert Multibox to Boxes

Operates on Multibox Objects.

```
MULTIBOXes will be divided into single BOXes.
Only BOXes may be modified.
```

---

94  Convert to Bounding Box

Operates on Geometric Objects.

```
Convert OBJECT(s) to their Bounding Box(es).
```

---

95  Convert to Poly

Operates on Solid Objects, Box Objects.

```
If possible, the selected Solid(s) will be transformed
into POLYGON(s).

Note: Only possible if a Solid is cylindrical in one direction


Convert BOX to extruded (closed) POLYgon.
Handles will be placed from edges to corners.
```

---

96  Convert to Solid

Operates on Geometric Objects.

```
Convert OBJECT(s) to Solid(s).

Note:
A Solid is an arbitrary 3D object consisting of triangles.
Handles will be placed at their corners.
```

---

97  Group Boxes

Operates on Box Objects.

```
Combines several BOXes into one MULTIBOX.
Multiboxes must have common TEXT and GROUP attributes.

Mutliboxes are needed in animations if the field
values have to be referred to a common maximum.
```

---

98  Split Solid

Operates on Solid Objects.

```
The selected solid will be divided into smaller parts.
Useful if multiple objects are grouped into one solid.
Undo will work.
```

Advanced/Edit/Optimize

---

99  Optimize Multibox

Operates on Box Objects.

```
Optimize discretized Object
```

---

100  Optimize Solid 1

Operates on Solid Objects.

```
try to repair the selected solid
```

---

101  Optimize Solid 2

Operates on Solid Objects.

```
try to optimize  the selected solid
```

---

102  Simplify Polygon

Operates on Polygon Objects.

```
Deletes obsolete points of a POLYGON to reduce the memory requirement
and increase drawing speed.

A point Pn between Pn-1 and Pn+1 is obsolete if it is
part of the line (Pn-1,Pn+1)
```

33.18.6. Advanced/Group

103  Copy to Current Group

Operates on Geometric Objects.

```
Copy the selected OBJECTs to the current (active) group.
```

104  Move to Current Group

Operates on Geometric Objects.

```
Move OBJECT(s) to the current group.
```

105  Select Groups' Objects

Operates on Geometric Objects.

```
Select all objects of the selected object's group
```

106  Set Height of Current Group

Operates on Geometric Objects.

```
The heights of selected OBJECTs are assigned to the active group's height.
```

33.18.7. Advanced/LibraryElements

107  Add Port Number(s)

Operates on Port Objects,operand Number Objects.
Shortcut: <plus>

```
The entered NUMBER will be added to the
Port number of the selected library element.
```

108  Explode Library Element

Operates on Port Objects.

```
Library Port:
The selected port is exploded into its
basic parts applying the current discretization,
i.e. metal boxes, current and voltage boxes.

Library Object:
The selected object is converted to a solid for
e.g. boolean operations with other solids.
Note: Parameters will be replaced by their values

The library element can only be recovered using Undo.
Selection: Library element
```

---

109 Show Exploded Library Element

Operates on Port Objects.

```
The selected library element is maintained. The
basic parts applying the current discretization,
i.e. metal boxes, current and voltage boxes are
shown in addition.

Use redraw to erase the exploded view.

Selection: Library element
```

---

110 Subtract Port Number(s)

Operates on Port Objects,operand Number Objects.
Shortcut: <minus>

```
The Port number of the selected library element
will be subtracted from the entered NUMBER.
```

33.18.8. Advanced/Linpolys & Bond Wires

---

111 Change to Line poly (Script)

Operates on Polygon Objects.

```
Create a bond wire between the start and end
point of the selected 2-point POLYGON.

Converts a two-point POLYGON into a set of points
delivered by a user-defined script.

A Python script called "bondwire.py" has to be
available in the project folder.
```

---

112 Create Bond Wire

Operates on Polygon Objects.

```
Create a bond wire between the start and end
point of the selected 2-point POLYGON.

The height and slope information is entered afterwards
or can be adjusted in the options menu: Bond wires.

A third-order polynomial will be used as the shape.
```

113 Create Line Poly

Operates on Polygon Objects.

```
Converts the selected POLY to LINPOLY.
The resulting LINPOLY is open.
By default, its tangent has zero components in the plane perpendicular
to the POLY plane.
```

114 Linpoly to Solid

Operates on Bondwire Objects,operand Number Objects.

```
Convert a LINPOLY into a SOLID

The cross section (diameter) is entered by a number.

Useful for 3D STL export
```

115 Set Tangential

Operates on Bondwire Objects,operand Arrow Objects.
Shortcut: <equal>

```
The entered ARROW defines the tangent of the LINPOLY.
The ARROW may not be entered in the Poly's plane.
Selection: ARROW and LINPOLY
```

33.18.9. Advanced/Mesh

Advanced/Mesh/AddLines

116 Add Fixed

Operates on Discretization Objects,operand Arrow Objects.
Shortcut: <asterisk>

```
The entered POINTs or ARROWs will be added as Fixed Lines
on the selected GRID bar.

Fixed Lines are marked black and bold on GRID bar.

To select press left mouse button on GRID bar.

Selection: POINT or ARROW and GRID
```

---

117  Add Fixed Lines

Operates on Discretization Objects,operand Point Objects.
Shortcut: <asterisk>

```
The entered POINTs or ARROWs will be added as Fixed Lines
on the selected GRID bar.

Fixed Lines are marked black and bold on GRID bar.

To select press left mouse button on GRID bar.

Selection: POINT or ARROW and GRID
```

---

118  Add To Discretization

Operates on Discretization Objects,operand Arrow Objects.
Shortcut: <plus>

```
The entered POINTs or ARROWs will be added as
un-fixed lines on the selected GRID bar.

Non-fixed Lines are marked as red thin lines.

To select press left mouse button on GRID bar.

Selection: POINT or ARROW and GRID
```

---

119  Add Unfixed Lines

Operates on Discretization Objects,operand Point Objects.
Shortcut: <plus>

```
The entered POINTs or ARROWs will be added as
un-fixed lines on the selected GRID bar.

Non-fixed Lines are marked as red thin lines.

To select press left mouse button on GRID bar.

Selection: POINT or ARROW and GRID
```

---

120  Arrow 1/3rd Rule

Operates on Arrow Objects,operand Number Objects.
Shortcut: <bar>

```
Generates two arrows according to the metallization rule.
This rule accounts for the edge effect by filling one
third of a cell with metal. The entered ARROW has
to be drawn between two edges of the metal object.
The length of the generated arrows is entered by
NUMBER and RETURN (=grid spacing).
The arrows can be used to define the grid lines by
activating the grid bar and pressing Add Fixed.

Note: Grid lines between the new ones have to be
deleted.
Selection: ARROW + NUMBER
```

---

121  Propose Discretization

Operates on Port Objects,operand Discretization Objects.
Shortcut: <plus>

```
Adds discretization lines to the selected GRID bar
proposed by the library element.

Note:
The rules for proposing lines can be adjusted with the library wizard.

Note: Use Explode, Discretize to check the right discretization.
Apply Undo to recover library element.

Selection: Library element and GRID bar
```

---

122  Propose Discretization (local)

Operates on Port Objects,operand Discretization Objects.
Shortcut: <minus>

```
Locally adds discretization lines to the selected
GRID bar proposed by the library element.

Note:
The rules for proposing lines can be adjusted
with the library wizard.

Note: Use Explode, Discretize to check the right discretization.
Apply Undo to recover library element.

Selection: Library element and GRID bar
```

Advanced/Mesh/DeleteLines

---

123  Delete Fixed Range

Operates on Discretization Objects,operand Arrow Objects.
Shortcut: <slash>

```
Delete all lines in the range defined by the entered ARROW
on the selected GRID bar.

Selection: ARROW and GRID
Undo will work.

Note:
A range of lines can also be deleted by dragging the
middle mouse button over the lines
```

---

124  Delete Mesh

```
Automatic and Manual meshing:
Deletes all lines (fixed and non-fixed) on all GRID bars.
```

---

125  Delete Range

Operates on Discretization Objects,operand Arrow Objects.
Shortcut: <minus>

```
Non-fixed gird lines will be deleted in the range defined
by ARROWs on the selected DISC bar.

To select press left mouse button on GRID bar.

Selection: POINT or ARROW and GRID bar

Note:
A range of lines can also be deleted by dragging the
middle mouse button over the lines
```

Advanced/Mesh/Edit

---

126  Copy Marked

Operates on Discretization Objects,operand Arrow Objects.
Shortcut: <percent>

```
GRID lines of marked intervals will be copied
according to the entered 1D ARROW.

The entered arrow acts as the shift vector.

Old grid lines will be maintained.
```

---

127  Fix Range

Operates on Discretization Objects,operand Arrow Objects.

```
All un-fixed (red) GRID lines covered by the entered
ARROW will be turned into fixed lines (black).

Fixed lines will be maintained.

Selection: GRID bar and ARROW
```

---

128  Shift Marked

Operates on Discretization Objects,operand Arrow Objects.
Shortcut: <numbersign>

```
Shift all marked lines according to the entered
ARROW on the selected GRID bar.

The entered arrow acts as the shift vector.

Selection: ARROW and GRID bar
```

---

129  Unfix Range

Operates on Discretization Objects,operand Arrow Objects.

```
Fixed lines (black) will be turned into
un-fixed (red) lines.

Un-fixed lines are maintained.

Selection: ARROW and GRID bar
```

Advanced/Mesh/ManualMesh

---

130  Generic Discretization

Operates on Discretization Objects.

```
Detects all edges of BOXes and corners of POLYs and fills
the selected GRID bar with Fixed Lines which are marked
 black and bold.

Note: Previous entered lines (also fixed ones) will be deleted.

Selection: GRID bar
Hint: More functionality is available by entering a NUMBER as base spacing.
Undo will work.
```

---

131  Smooth Discretization

Operates on Discretization Objects.

```
All Intervals on the selected GRID bars are filled automatically
with non-fixed lines using a special smoothing algorithm.

An interval must be enclosed by a Pair of 2 Fixed Lines
(entered by e.g. middle mouse button on the grid bar)

The algorithm can be used to obtain a smooth transition
from fine to coarse mesh and vice versa.

Selection: GRID bar
Note: Non-fixed lines will be deleted.
Undo will work.
```

---

132  Smooth Discretization Marked

Operates on Discretization Objects.

```
Only the marked intervals on the selected GRID bars are filled automatically
using a special smoothing algorithm.

An interval must be enclosed by a Pair of 2 Fixed Lines.

The algorithm can be used to obtain a smooth transition from
fine to coarse mesh.

Note:
To mark an interval press (or drag) left mouse button on GRID bar in the
respective intervals

Note: Non-fixed lines will be deleted.
Undo will work.
```

Advanced/Mesh/Marks

---

133  Delete Marks

Operates on Discretization Objects.

```
Deletes all marks on the selected GRID bar.

Note:
Separate marks can be deleted by clicking
the mark with the left mouse button.
```

---

134  Mark All

Operates on Discretization Objects.

```
Inserts marks between every two grid lines on the selected GRID bar.

Note:
Separate marks can be inserted by clicking the mark with the left mouse button.
```

---

135  Mark Range

Operates on Discretization Objects,operand Arrow Objects.
Shortcut: <dollar>

```
All intervals between GRID lines covered by the
entered ARROW will be marked.

Useful for mesh refinement in certain regions,
e.g. enter 2 followed by / adds one grid line
in each marked interval.
```

33.18.10. Advanced/Parameters

136  Move Parameter

Operates on Geometric Objects.

```
The selected OBJECT(s) will be moved by a parametric value.

Note:
A parametric value is defined by entering a textstring for a
coordinate value and defining range and step size.
```

137  Set Parameters by File

Operates on Port Objects.

```
The File consists of lines

x=A y=B
x=C y=D

Each line will be applied to change the parameters of one selected
object. The selected objects are sorted top-bottom and left-right.

If the file's ending is .py, it is interpreted as a python script with a
function "parameters(i,x,y)",  getting the object's Number
(starting with zero) and projected coordinates. The function returns
one line of  the file defined above.
```

138  Stretch Parameter

Operates on Geometric Objects.

```
Edges or corners of selected OBJECT(s) defined by the
active (green) handles will be stretched by a parametric
value.

Note:
A parametric value is defined by entering a textstring for
a coordinate value and defining range and step size.
```

33.18.11. Advanced/Polygons

139  Bisect Polygon

Operates on Polygon Objects.

```
The selected POLY is split in two along a line defined
by two activated (green) handles.

Note:
Exactly two handles must be active (green).

Selection: POLY
```

---

140  Circle Recognition

Operates on Polygon Objects,operand Number Objects.

```
Polygons resembling circles/cylinders will be recognized and
converted to (1-point) cylinder objects.

This is often needed for imported layout data which contains
cylinders in the form of multiple point polygons.

The circle recognition outcome depends on the following
options in "Editor Options - Components - Polygon":
- Circle Recognition Points
- Circle Recognition Tolerance
```

---

141  Cycle Elements

Operates on Polygon Objects,operand Number Objects.

```
Start and end point of the selected POLYGON(s) will be
cycled according to the entered Number.
```

---

142  Poly to Rounded Strip

Operates on Polygon Objects,operand Number Objects.

```
Create a stripline from a path defined by a polygon.

Converts the selected POLY into a LINPOLY with a rectangular
cross section with a width entered by the number and a height
defined by the layer arrow.

All corners will be mitred with the diameter of the width size.
```

---

143  Poly to Strip

Operates on Polygon Objects,operand Number Objects.

```
Create a stripline from a path defined by a polygon.

Converts the selected POLY into a LINPOLY with a rectangular
cross section with a width entered by the number and a height
defined by the group arrow.
```

---

144  Poly to Wire

Operates on Polygon Objects,operand Number Objects.
Shortcut: <numbersign>

```
Create a wire from a path defined by the selected Polygon.

Converts the selected POLY into a LINPOLY with a circular
cross section with a diameter entered by A NUMBER.

Note:
The center of the wire will be placed at the Polygon bottom.
Note:
The diameter has to be resolved by at least 4 cells in
perpendicular direction. Therefore, the NUMBER has to be
larger than 4 cells.

Example: Polygon (0-1000), Diameter 100, requires at least
         4 grid lines, e.g. at -50,-30,0,30,50

Selection: POLY and NUMBER
```

---

145  Rasterize Polygon

Operates on Geometric Objects.

```
The corners and edges of selected OBJECT(s) are mapped
to a grid which is defined by the parameters defined in:

Options - Poly - Poly Rasterize

Note:
This function is often used to get rid of odd co-ordinate
values of imported layout data.
```

---

146  Set Radius

Operates on Polygon Objects,operand Number Objects.

```
The activated (green) corners of the selected POLYGONs
will be rounded according to the entered NUMBER
which defines the radius of the rounding.

To activate/de-activate the corners press the left mouse
button.
```

33.18.12. Advanced/Properties

---

147  Add Property

Operates on Geometric Objects.

```
Assigns an additional Property to the selected OBJECT.

The Property string is composed by the Property Editor.
```

---

148  Delete Last Property

Operates on Geometric Objects.

```
Deletes the last Property entry which was bound to the selected object.
```

---

149  Delete Properties

Operates on Geometric Objects.

```
Deletes all Properties which are bound to the selected OBJECT.
```

---

150  Edit Properties

Operates on Geometric Objects.

```
Assigns a Property to the selected object using the Property editor.

Note: Property of the group will be ignored!
```

---

151  Set Property

Operates on Geometric Objects,operand Text Objects.
Shortcut: <equal>

```
Sets the current Property string to the object
Selection: OBJECT and TEXT
```

33.18.13. Advanced/Scripting

152  User Vertex Transform

Operates on Solid Objects.

```
Vertices of the selected Solid(s) will be transformed
according to a user defined Python script.

Note: A Python script .py has to be located in the project folder
```

33.18.14. Advanced/Select

153  Deselect Enclosed (D)

Operates on Arrow Objects,operand Geometric Objects.
Shortcut: D

```
De-Selects all objects which are enclosed by the entered ARROW.

Note:
Objects on a locked or switched-off layer are disregarded.
```

154  Deselect Overlapping (d)

Operates on Arrow Objects,operand Geometric Objects.
Shortcut: d

```
De-Selects all objects which are enclosed or partially lie
inside the region defined by the entered ARROW.
```

155  Select Outside

Operates on Arrow Objects.

```
Selects all objects which are outside of the
region covered by the entered ARROW.

Note:
Objects on a locked group are disregarded.
```

33.18.15. Advanced/Solid

156  Extrude Flat Solid

Operates on Solid Objects,operand Number Objects.

```
The selected Solid(s) will be extruded in direction of its normal.
The thickness of this extrusion is defined by the entered number.

Selection: OBJECT and NUMBER

Hint:
Useful for unclosed Solids with zero thickness
```

33.18.16. Advanced/Transform

157  Scale to Object Center

Operates on Polygon Objects, Box Objects,operand Number Objects.

```
The selected BOX(es) will be scaled according to
the entered NUMBER which acts as a scale factor.

The scaling is referenced to each object's center.
```

158  Scale to the Origin

Operates on Solid Objects, Geometric Objects,operand Number Objects.

```
The selected OBJECT(s) will be scaled according to
the entered NUMBER which acts as a scale factor.

The scaling is referenced to the co-ordinate origin
so that all co-ordinate values are multiplied by
the scale factor.
```

159  2D Scale to Origin

Operates on Geometric Objects,operand Number Objects.

```
The selected OBJECT(s) will be scaled according to
the entered NUMBER which acts as a scale factor.

The scaling is referenced to the co-ordinate origin
so that all co-ordinate values are multiplied by
the scale factor.

The Object is scaled in the actual drawing Plane. The perpenicular geometry is unchanged.
```

---

160  Align Object to Arrow

Operates on Geometric Objects,operand Arrow Objects.

```
The midpoint of the selected BOXes will be moved
to the center of an arrow

Selection: BOXes and Point
```

---

161  Align Object to Point

Operates on Geometric Objects,operand Point Objects.

```
The midpoint of the selected BOXes will be centered
to the entered Point

Selection: BOXes and Point
```

---

162  Oversize

Operates on Polygon Objects, Box Objects, Solid Objects,operand Number Objects.

```
The selected OBJECTs will be oversized according to the entered NUMBER.
All edges of the OBJECT are shifted to the outside.

Selection: OBJECT and NUMBER
```

---

163  Oversize with Roundings

Operates on Polygon Objects, operand Number Objects.

```
The selected OBJECTs will be oversized according to the entered NUMBER.
All edges of the OBJECT are shifted to the outside with circular miters.

Selection: OBJECT and NUMBER
```

---

164  Rotate Structure and Mesh

Operates unconditionally.

```
To be executed in the 2D Design Mode. Rotates structure and
mesh in 90 Degrees steps
```

34. Editor Options

34.1. 2D Design

34.1.1. 2D Design/Arrow

---

1 Arrow color
Default *red*

Should be different to background color

```
Select color from list or enter a known color name,
e.g. magenta, light green, dark blue, grey20
```

---

2 Arrow Rectangle Color
Default *grey25*

Fill color of area which is spanned by the entered arrow

```
Select color from list or enter a known color name,
e.g. magenta, light green, dark blue, grey20
```

---

3 Arrow Text
Default *show distance*

Change arrow text display

```
off:  only arrow and rectangle
normal:  arrow number
verbose:  arrow number and coordinates
```

34.1.2. 2D Design/Component Snap

---

4 Snap Mode
Default *components+traces*

Cursor snap on components

---

5 Horizontal Component Snap
Default *2*

Cursor snap on component lines

Used for selections, Distance in Pixels

---

6 Vertical Component Snap
Default = *horizontal*

Used for selections, Distance in Pixels

---

7 Snap on distant Components
Default *on*

Cursor snap on x,y,z coordinates of Object lines

Cursor snap on x,y,z coordinates of Object lines

34.1.3. 2D Design/Cursor

---

8 Cursor Color
Default *grey20*

Color of crosshair and cursor numbers

Select color from list or enter a known color name,
e.g. magenta, light green, dark blue

---

9 Number of Digits
Default *2*

Adjust cursor co-ordinate display

By default 2 digits after the decimal point are used
to display the cursor co-ordinate.

Hint:
If a question mark is displayed after the last digit, the
current coordinate is a rounded number, only

---

10 Cursor Size
Default *Big*

Length of the cursor crosshair

\newline Default = Big displays the cursor entirely over the drawing area

---

11 Show Mesh Coordinate
Default *off*

Switch cursor display mode

The discrete coordinates are available if a
mesh has been generated.

A discrete co-ordinate is the number of mesh line
counted from the mimimum boundary.

34.1.4. 2D Design/General Snap

12 Snapped Cursor Color
Default *#f70*

Cursor color changes

```
In case of

- grid lines are hit
  the cursor co-ordinate values are highlighted
- object lines are hit
  the crosshair is highlighted in this color
```

13 Horizontal Snap
Default *small*

Behavior of the cursor snap

```
off           no snap on co-ordinates
small        fine auto snap, depending on zoom
medium    medium auto snap, depending on zoom
big           coarse auto snap, depending on zoom
value       snap at horizontal co-ordinates N*value
```

14 Vertical Snap
Default = *horizontal*

Behavior of the cursor snap

```
horizontal  - use same setting
off           no snap on co-ordinates
small        fine auto snap, depending on zoom
medium    medium auto snap, depending on zoom
big           coarse auto snap, depending on zoom
value       snap at horizontal co-ordinates N*value
```

15 Horizontal Mesh Snap
Default *off*

Snap on mesh lines

```
If turned on, crosshair snaps to gridlines and changes
color to red.
```

16 Vertical Mesh Snap
Default = *horizontal*

Snap on mesh lines

```
If turned on, crosshair snaps to gridlines and changes
color to red.
```

34.1.5. 2D Design/Point

---

17 Point color
Default *red*

Should be different to background color

```
Select color from list or enter a known color name,
e.g. magenta, light green, dark blue, grey20
```

---

18 Point size
Default *30.000*

Size of Point and Arrow crosses

```
Size in Pixels
```

---

19 Point Text
Default *show point number*

Change Point text display

```
off:  only crosshair
normal:  point number
verbose:  point number and coordinates
```

34.1.6. 2D Design/View

---

20 Background Color
Default *#DFDFF0*

---

21 Group Drawing Order
Default *normal*

Specifies drawing order in 2D Design mode outlined view

```
In the outlined view (last top group icon) objects are drawn in a filled style.
Here, it can be specified which group is on top.

- Normal: First group =  Background, last group = Foreground
- Reverse: First group = Foreground,  last group = Background
```

---

22 Wheel Zoom Factor
Default *1.2*

Enlargement / Reduction factor of one wheel turn

```
Draft mode

wheel backward = zoom out
wheel forward = zoom in
```

---

23 Display Mesh
Default *off*

Display mesh lines in drawing area

```
By default "off" no mesh lines are visible inside the
drawing area. Instead they are displayed on the
grid bars on the right and in the bottom.

Displaying can be enabled by selecting one of the colors

Press "Redraw" to update the window
```

---

24 Display Grid
Default *off*

Display grid lines in drawing area

```
By default "off" no grid lines are visible inside the
drawing area. Instead coordinates are displayed at
the cursor cross hair.

Displaying can be enabled by selecting one of the colors

Press "Redraw" to update the drawing
```

34.2. 2D Graphs

---

25 Max. Number of curves with full table display
Default *200*

Max. Number of curves with full table display

```
Maximum number of curves displayed in the data table with a full set of parameter columns.

If this number is exceeded, all parameters are combined in the same column to retain GUI performance.
```

34.3. 3D Design

---

26 Min. Normal Angle for Wireframe
Default *30*

Threshold for displaying lines of solids

```
In draft mode all objects are displayed as wire
frame models. Solid objects contain many lines
which can hide other objects.

To reduce the number of lines displayed in
draft mode, the minimum angle can be increased.
```

---

27 Max. Number of Handles
Default *1000*

Max. Number of Hanldes per Object

34.4. 3D Display

34.4.1. 3D Display/Field Monitor Animation

---

28 Speed in ms
Default *1000*

Adjust speed of animation

```
This number can be adjusted if animations loops
run too slow or too fast by reducing or increasing
this number, respectively.
```

---

29 Linewidth
Default *3*

Width of grid- or contour lines in animations

---

30 Pointsize
Default *10*

Size of points in animations

```
Define the size of field monitor animations in 'points' mode.
```

34.4.2. 3D Display/Lights

---

31 Light 0 Position
Default *0 0 1 0*

Adjust point light source

```
Light Positions
1000  x= inf
-1000  x= -inf
0100  y= inf
0-100  y= -inf
0010  z= inf
00-10  z= -inf
0001  off
```

---

32 Light 0 Ambient
Default *0.2*

Pointless light source

---

33 Light 0 Diffuse
Default *0.4*

Diffuse reflection coefficient

---

34 Light 0 Specular
Default *0.9*

Specular reflection coefficient

---

35 Light 1 Position
Default *off*

See Light 0

`Optional additional light source`

---

36 Light 1 Ambient
Default *1.0*

See Light 0

---

37 Light 1 Diffuse
Default *0.2*

See Light 0

---

38 Light 1 Specular
Default *0.0*

See Light 0

---

39 Light Global Ambient
Default *0.2*

Pointless light source

`Brightness is enlarged if number is increased.`

34.4.3. 3D Display/View

---

40 Background Color
Default *white*

Color can be changed at any time

---

41 Background Gradient
Default *smooth_a*

Background Gradient

```
Gradient algorithms for 3D Background

Hint:
Switch to "Off" if a white background should be obtained
```

---

42 Display Mode
Default *fill*

Display style of objects in 3D mode

```
Either Shaded or Wire frame model
```

---

43 Resolution
Default *auto*

Resolution of the 3D display in Pixel

```
e.g. for Icon creation set to 24x24
```

---

44 Transparency Setup
Default *accurate hw*

Option depending on graphic card support

```
For proper display of transparent media switch to accurate.
+ accurate hw   fastest, not supported on all graphic cards
+ accurate hw/sw  fast, not supported on all graphic cards
+ accurate sw  slow, should work in any case
```

---

45 Software 3D antialiasing
Default *off*

Improve rendering

```
If enabled an antialiasing algorithm tries to improve the
appearance of the rendered image.
```

34.5. Components

34.5.1. Components/Bond Wires

---

46 Slope Angle phi0=
Default *50.0*

Anlge at start of bond wire

```
0 < phi0 < 90 degree
```

---

47 Slope Angle phi1=
Default *50.0*

Anlge at end of bond wire

```
0 < phi1 < 90 degree
```

---

48 Number of Points in Bond Wire
Default *10*

Increase this number to obtain a smoother wire

```
N > 2
```

---

49 3D Display Linewidth
Default *5*

Reduce this number if the appearance is too bold in 3D mode

```
Width in number of Pixels
```

34.5.2. Components/Polygons

---

50 Poly Rasterize
Default *0.1*

Set raster width

```
The corners and edges of selected OBJECT(s) are mapped
to a grid which is defined by this number.

Note:
This function is often used to get rid of odd co-ordinate
values of imported layout data.
```

---

51 Poly Minimum Angle
Default *0.01*

Set min. simplification angle

The minimum angle for poly simplification.

---

52 Circle Resolution (Degrees)
Default *11.25*

Number of points used to approximate a circle

Usually a circle consists of the data set Midpoint (x0,y0) and Radius r0. If the circle is merged of subtracted from other objects the cirlce resolution defines the arc which is approximated by a line.

Note:
If only a small circle segment is needed, reduce the circle resolution to e.g. 1 degree

---

53 Rotpoly Subdivision (Integer)
Default *36*

Number of segments used to approximate a body of revolution

To save drawing time a rotational polygon is displayed in segments. The more segments are used the better is the approximation

Note:
If only a small part of the body needed, increase the number of segments to e.g. 360

---

54 Automatic Circles
Default *off*

Switch on circle recognition at import/export merge etc.

---

55 Line Poly
Default Width
Default *0*

Number of cells for discretization

N = 0 - effective diameter appr. 0.2 * delta
N = 1 - effective diameter appr.   1 * delta

---

56 Poly Set Radius Option
Default *SET*

Options for poly/ set radius

CW: set only clockwise (an selected) vertices
CCW: set only CCW vertices
SMOOTH: smoot adjacent bows with new radius

Components/Polygons/Advanced

---

57 Poly tolerance
Default *0.0001*

Tolerance in units

```
Used in the following operations:
- Path to Poly
- usertransformation
- oversize
- discretize
```

---

58 Rotpoly Fast Discretization
Default *off*

Accelerate Meshing of rotational polygons

---

59 Show Bounding Box
Default *2d perpendicular*

Display bounding box of polygons

---

60 Circle Recognition min. # Points
Default *10*

Simplify polygons

```
Objects can be simplified to (1-point) circles to improve
import and drawing speed.

Usage:
- Select polygons to be simplified to 1-point circles
- Press "Advanced - Circle recognition"
- Enter tolerance

Note:
To prevent octagons to be recognized as circles
decrease this number.
```

34.5.3. Components/Solids

---

61 Min. Normal Angle in 2D display
Default *30*

Threshold for displaying lines of solids

```
In 2D Design mode all objects are displayed as wire
frame models. Solid objects contain many lines
which can hide other objects.

To reduce the number of lines displayed in
2d Design mode, the minimum angle can be increased.
```

---

62 Max Number of Solid Handles
Default *10000*

Threshold for displaying handles of solids

```
The selection of solids can take several time
depending on the number of handles which
are displayed at each corner of a triangular
subelement.

This number can be enlarged if all handles
should be displayed.
```

---

63 Warn for Improper Objects
Default *on*

Problems when importing solids

```
Only solid objects which surface is closed can be
properly processed.
An un-closed solid can be healed by using the operations
"Extrude Solid" or "Orient Solid"

Note:
If a solid object is not closed, some operations can not
be performed.
```

---

64 Save all Triangles in ASCII
Default *off*

Save all triangles as ASCII in gym file

Components/Solids/Advanced

---

65 Relative Accuracy
Default *1e-5*

Used for Solid operations

---

66 2D Draft Display
Default *all*

Accelerate display by neglecting triangles

34.5.4. Components/Via Fence

Components/Via Fence/Geometry

---

67 Distance to Poly
Default *800*

Distance between Polygon and Pad

```
If distance > 0 the via fence will be created outside the polygon
```

---

68 Via Diameter
Default *200*

Should be smaller than Pad Width

---

69 Via Distance
Default *500*

Maximum Distance

```
The distances between vias are determined by an iterative procedure
```

---

70 Pad Width
Default *300*

Should be larger than Via Diameter

Components/Via Fence/Groups

---

71 Via A Group(s)
Default *Via*

Name of group(s) where vias are placed

```
If Via A and Via B names are different, alternating vias will be
created, e.g. to create staggered vias:
Via 1,3,5,7,... in Via A group
Via 2,4,6,8,... in Via B group

Note:
- If group is already defined vias will be created in this group
- If group name is not already defined, the group will be created
- If more than 1 name is entered, vias will be created in several groups

Hint:
- Check group height and property after creation
```

---

72 Via B Group(s)
Default *Via*

Name of group(s)

```
If Via A and Via B names are different, alternating vias will be
created, e.g. to create staggered vias:
Via 1,3,5,7,... in Via A group
Via 2,4,6,8,... in Via B group

Note:
- If group is already defined vias will be created in this group
- If group name is not already defined, the group will be created
- If more than 1 name is entered, vias will be created in several groups

Hint:
- Check group height and property after creation
```

---

73 Pad Group(s)
Default *Pad*

Name of Group(s)

```
Note:
- If group is already defined vias will be created in this group
- If group name is not already defined, the group will be created
- If more than 1 name is entered, pads will be created in several groups

Hint:
- Check group height and property after creation
```

Components/Via Fence/Options

---

74 Via # Points
Default *16*

Circle resolution

```
Use, e.g. 4 for square vias
```

---

75 Use slow DRC check
Default *off*

---

76 Maximum Iterations
Default *1000*

---

77 Use Attractors
Default *on*

Set vias in corner centers

```
Corner centers act as attractors for the via filling
algorithm
```

---

78 DRC Failure Behavior
Default *error*

---

79 Logging
Default *window*

34.6. Design

---

80 Design Check Tolerance
Default *0.010*

DRC Tolerance in drawing units

```
Distances below this limit will be disregarded
```

---

81 Rotation Angle
Default *90*

Constructive and destructive point rotation

```
positive: counter clockwise
negative: clockwise

Note:
Rotation axis = line through point parallel to normal direction
           of current draing plane

Multiple angles are possible for constructive rotation e.g. 90@3 makes
3 copies.
```

---

82 Minimum Auto Thickness
Default *0.001*

Thickness used for Display in 3D mode

```
Some objects may be mapped on one single gridline,
e.g. metal strips. This value determines the minimum
thickness which is used for discretizing and displaying
these objects.

Can be reduced in case of discretization errors

Note:
Using a smaller drawing unit (e.g. um instead of mm)
is usually recommended in case of small extensions.
```

---

83 Multiple Copy Expression
Default *mcopy(dx=100,dy=100,nx=5,ny=5)*

Multiple copy

```
Copy selected OBJECT(s) according to an array expression:

mcopy(dx=a,dy=b,dz=c,nx=m,ny=l,nz=p,px=A,py=B,pz=C)

where a,b,c are the shifts in x,y,z direction
A,B,C are port-number-shifts in x,y,z direction
and  m,l,p are the numbers of copies in x,y,z direction
```

34.7. Dimensioning

---

84 Dimensioning Number of Digits
Default *0*

Display of dimensioning numbers in Draft mode

```
Usage:

- Enter starting point of dimensioning line
- Enter point of first object's edge
- Enter point of second object's edge
- Press "Create Measure"
```

---

85 Dimensioning Scale Factor
Default *1*

By default (1) the numbers are displayed in the selected unit.

```
Scale factor multiplies the displayed dimensioning value

Hint:
Press redraw to update numbers
```

---

86 Dimensioning Tic Size
Default *5*

Distance of dimensioning numbers to lines in Draft mode

```
Effects on dimensioning line

Hint:
Press redraw to update line display
```

---

87 Arrow Distance
Default *4*

---

88 Arrow Heads
Default *AUTO*

Shape of labelling arrows in Draft mode

```
a b c

a  tip style
b  Length of tip
c  width of tip

Hint:

Press redraw to see effect
```

---

89 Font Size
Default *12*

Size of labelling characters in Draft mode

---

90 3D Measure Cone size
Default *50*

Size of tip in 3D mode

---

91 3D Measure Font size
Default *14*

Size of Font in 3D Mode

---

92 3D Measure Line Width
Default *2*

Line width in 3D mode

---

93 3D Measure Distance
Default *100*

Distance between Line and Number

34.8. Export

34.8.1. Export/2D Image

---

94 Resolution Factor
Default *1*

Enlargement factor for 2D design images

```
Print creates an image from the current 2D design
```

34.8.2. Export/2D Layout

Export/2D Layout/DXF

---

95 DXF Scale
Default *1*

Adjust output unit

```
If the drawing unit is micron then the scale of 1
generates an output of micrometers.
```

---

96 Circle Tolerance
Default *0.01*

```
Circle recognition tolerance (relative)
```

---

97 Circle Recognition min. # Points
Default *12*

```
Minimum number of points for dxf circle recognition
```

---

98 Arc Recognition min. # Segments
Default *0*

```
Minimum number of segments for dxf arc recognition

0 switches arc recognition off.
```

Export/2D Layout/GDS

---

99 GDSII Scale
Default *1*

Adjust output unit

```
Usually, GDS input is on a very fine scale, such as nanometers.

If the drawing unit is micron then the scale of 1000
generates an output of nanometers.
```

Export/2D Layout/GERBER

---

100 GERBER Scale
Default *1*

Adjust output unit

```
Usually, gerber  input is in millimeters.

If the drawing unit is micron then the scale of 0.001
generates an output of millimeters.
```

---

101 GERBER Aperture Size (mm)
Default *0.001*

---

102 GERBER Float Format
Default *34*

34.8.3. Export/3D Image

---

103 Resolution Factor
Default *1*

Image enlargement factor

```
By default the generation of images of the 3D display
mode is done in the monitor resolution.

The resolution can be enhanced (e.g. for high quality
pictures to be used for posters) by increasing this number.
```

---

104 Show Coordinate System
Default *on*

Show Coordinate System in 3d image export

34.8.4. Export/3D Solids

---

105 STL Scale
Default *1*

Adjust output unit

```
If the drawing unit is micron then the scale of 1
generates an output of micrometers.
```

---

106 Separate STL Files
Default *off*

Importing multiple solids

```
Generate one stl file for each object.
```

34.8.5. Export/Advanced

Export/Advanced/Export EXCELLON

---

107 Start Coordinates
Default *0 0*

---

108 Unit
Default *mm*

---

109 Scaling Factor
Default *1./1000*

---

110 Output Format
Default *%.3f*

---

111 Circle min # points
Default *12*

---

112 Circle tolerance
Default *10*

Export/Advanced/Export HPGL

---

113 HPGL scale
Default *1/25.4*

Export/Advanced/Export MLS

---

114 Start Coordinates
Default *0 0*

---

115 Scale Factor
Default *1.0*

---

116 Repetitions
Default *1*

---

117 Undersize
Default *35*

---

118 Max Circle diameter
Default *6000*

---

119 Double Via Holes
Default *on*

---

120 Circle min # points
Default *12*

---

121 Circle diameter Scale
Default *160.0/58.0*

---

122 Circle tolerance
Default *10*

---

123 Poly Tolerance
Default *10*

---

124 Laser off speed
Default *200*

---

125 Laser on speed
Default *1*

---

126 Galvo speed
Default *1*

---

127 Max instructions per file
Default *200*

---

128 Subroutine Optimization
Default *on*

---

129 Lamp Current (A)
Default *19.5*

---

130 Pulse Frequency (kHz)
Default *1.5*

---

131 Pulse Length (s)
Default *1.5*

Export/Advanced/Export PS

---

132 POSTSCRIPT Page Width
Default *280 mm*

---

133 PS Layout Scale
Default *72.0/25400.0*

---

134 PS Layout shift
Default *10000*

Export/Advanced/Export UHT

---

135 Coordinate Shift
Default *0 0*

34.9. Import

34.9.1. Import/2D Layout

---

136 Import Scale Factor
Default *1*

Import scale factor

`Adjust if scale is different`

---

137 Arc Resolution (DEG)
Default *5*

Subdivision of rounded objects for DXF/DSN/GBR/EXC

```
By default (5 degree) a ciclre is divided into
72 points. In case of a huge amount of single
circles this number can be increased to accelerate
the import.
```

---

138 Accept Zero Area Objects
Default *off*

---

139 Circle Recognition Points
Default *10*

Circle Recognition minimum number of points

```
Objects can be simplified to (1-point) circles to improve
import and drawing speed.
```

---

140 Circle Recognition Tolerance
Default *0.1*

Simplify polygons

```
To recognize an object to be a cirlce the relative deviation
from a circular shape may not be larger than the value.
```

Import/2D Layout/DXF

---

141 DXF Connection Tolerance
Default *0*

Join single line ends

```
In some cases imported polygons consist
of single lines which can be joined by setting
the connection tolerance greater than zero.
```

---

142 DXF General Tolerance
Default *1e-3*

Absolute value in units

```
If the difference between two co-ordinate values are
below this threshold they will be considered as identical.

Note:
Decrease this limit if necessary or use an import scale
of 1000 (recommended)
```

---

143 Ignore layer in DXF block
Default *off*

Import/2D Layout/GDSII

---

144 GDS VIA Size (GDS units)
Default *0*

---

145 GDS VIA Distance (GDS units)
Default *1000*

---

146 GDS min. Obj size (GDS units)
Default *0*

Import/2D Layout/Gerber

---

147 Merge GERBER Layers
Default *on*

Merge gerber file's objects

```
Merge all objects in a gerber file before import. May be slow/memory
consuming for big structures.
```

---

148 GERBER Merge Tolerance
Default *1e-06*

Tolerance for GERBER merger, in screen units.

```
Line ends which distances are below this limit will be connected
```

34.9.2. Import/3D Solids

---

149 Separate STL Objects
Default *off*

Importing multiple solids

```
By default (off) one imported binary STL file generates one layer
and one solid object.

If the STL file contains several single objects they can be
seperated on multiple layers by switching to (on).
```

---

150 STL Shift (x y z)
Default *0 0 0*

Insertion point of imported objects

Adjust if origin is different

---

151 STL Simplification
Default *0*

Simplify 3D objects

Accelerate import of complex 3D structures by joining surface triangles

Number > 0: Angle

Neighbouring triangles which normals differ less this angle will be joined

Number < 0: Distance

Neighbouring triangles which size are less this magnitude will be joined

34.9.3. Import/Empire

---

152 GYM Scale Factor
Default *1*

Import scale factor, for gym files.

Adjust if scale is different

---

153 GYM Shift (x y z)
Default *0 0 0*

Insertion point of imported objects

Adjust if origin is different

34.9.4. Import/Advanced

---

154 EXCELLON decimal digits
Default *4*

---

155 HPGL pen width
Default *300*

34.10. Preferences

156 Create Backup Files
Default *on*

Backup files are named file.gym.bak

```
On every "Save" action the backup file will be updated

To recover backup files, rename file.gbk to file.gym
```

157 Icon Size
Default
Default

Button Appearance

```
By default all Operation buttons (icons) have a size
of 24x24 pixels and are optimized for a screen of
1280x1024 size.
```

35. Autodisc Options

35.1. General

---

158 Resolve Curved Geometry
Default *no*

Helps to resolve curvatures and Tapers

```
Increase mesh resolution in areas with round geometry
```

35.2. Advanced

35.2.1. Advanced/Options

---

159 Mesh Resolution (Numeric)
Default *5*

Map resolution quality to number

```
-1 -> Very coarse,
1->coarse,
5->medium,
10->fine etc.

Can be defined as parameter, e.g $res for automatic convergency checks
```

---

160 More Lines between Edges
Default *no*

For experimental use can be switched on

```
Can help to resolve thin dielectrics with curvatures
```

---

161 Flat Metal Edge correction
Default *yes*

For experimental use can be switched off

```
Use the 1/3-2/3 flat metal rule
```

---

162 Object Scan Angle (Degrees)
Default *30*

Minimum Angle for object scan interval detection

```
Minimum Angle of segments against actual direction
```

---

163 Relative Drawing Accuracy
Default *1e-6*

```
Two object lines are considered to have equal
co-ordinates if their distance are below this
relative limit.

The absolute limit is determined by this value
times (max-min), which is the distance between
the boundaries in the respective direction.

Increasing this value can help to avoid grid lines
which are very close.

Note: This function works for autmatic meshing, only.
```

---

164 Relative Limit for Warning
Default *1e-3*

Minimum cell spacing

```
Very small cell spacing should be avoided if possible
to reduce simulation time.

A warning message is displayed if a cell is detected
which is smaller than 0.001*(max-min) by \newline Default,
wehre (max-min) is the distance between the
boundaries in the respective direction.

If these small grid line spacing is intended the
warning messages can be avoided by reducing
this number.
```

---

165 Object Roughness (cells)
Default *0*

Accounts for drawing inaccuracy

```
Number in Units can be adjusted for improper drawings.
```

35.2.2. Advanced/User Setup

---

166 Min Cells per Wavelength
Default *10*

Changeable if Discretization set to User defined

```
Spatial sampling of waves

- Minimum: 6 cells / wavelength
- Recommended Minimum: 10 cells / wavelenth
- Reduce numerical dispersion: 20 cells / wavelength

Note:
Wavelength is derived from end frequency
```

---

167 Absolute min in units
Default *0*

Changeable if Discretization set to User defined

```
Distances below this limit will not be recognized for meshing.
May be necessary to change if large units are selected.

Format:
d                  Absolute minimum resolution
dx_dy_dz     Absolute minimum resolution in
                    each direction
```

---

168 Target min in units
Default *50*

Changeable if Discretization set to User defined

```
The meshing algorithm will try to aim for this desired
minimum resolution.

Format:
d                  Desired minimum resolution
dx_dy_dz     Desired minimum resolution in
                   each direction
```

---

169 Max Resolution in units
Default *1000*

Changeable if Discretization set to User defined

```
None:             Maximum resolution is determined by upper frequency
d:                  Maximum resolution
dx_dy_dz:    Maximum resolution in respective direction
```

---

170 Arc Resolution in Degrees
Default *12.125*

Changeable if Discretization set to User defined

```
Value determies the angular step which
is used to discretize arcs and circles.
```

---

171 Planar Direction
Default *0*

Changeable if Discretization set to User defined

If Automesh Setup - General - Mode is set to Planar
this value determines the planar plane.

---

172 Object cells 1
Default *5*

Changeable if Discretization set to User defined

Value determines:

- Number of cells for gap to the boundaries
- Number of cells used to resolve lumped objects
- Number of grid lines if flat metal discretization is set to 1 cell

---

173 Object cells 2
Default *5*

Changeable if Discretization set to User defined

Value determines:

Number of cells used for basic discretization of objects
(if group discretization is set to 'basic', no 1/3rd rule)

---

174 Object cells 3
Default *5*

Changeable if Discretization set to User defined

Currently not in use.

---

175 Object cells 4
Default *2*

Changeable if Discretization set to User defined

Value determines:

Number of cells used to resolve important objects
(If group meshing level is set to 'important')

---

176 Refinement ratio
Default *1.5_1.5_3*

Changeable if Discretization set to User defined

Factor for graded meshing, used for the
smooth transition from small cell spacing
to large cell spacing.

\newline Default: d(i+1) = 1.8 * d(i)

Can be set for each direction separately, e.g.
1.1_1.4_1.8

---

177 max. Aspect ratio
Default *5*

Changeable if Discretization set to User defined

```
Maximum desired Aspact ratio between directions based on bounding-boxes.
```

---

178 Edge accuracy (Percent)
Default *50*

Changeable only if Discretization set to User defined

```
Used for the decision to merge grid lines.

If two grid lines with distance dp are located
ds from a visible edge they will be merged if

dp < k/100 * ds, where k is the edge accuracy in percent
```

---

179 Flat metal Recognition Limit
Default *0.05*

Changeable if Discretization set to User defined

```
A metal object is considered to be flat
if the ratio between thickness and
cross section is below this limit.
```

---

180 Max. scan points per Object
Default *1000*

Changeable if Discretization set to User defined

```
Maximum Number of scan points a single Object can create.
```

---

181 Wedge limit in Degrees
Default *15*

Changeable only if Discretization set to User defined

```
Detection of inner edges in polygons and solids
```

35.3. Axis Setup

---

182 x mesh in units
Default *auto*

```
off:     Omit this direction for automatic meshing
auto:     automatic meshing
auto+fixed:    automatic meshing; keeping fixed lines
(number):    equidistant meshing with spacing of (number)
```

---

183 y mesh in units
Default *auto*

```
off:      Omit this direction for automatic meshing
auto:      automatic meshing
auto+fixed:     automatic meshing; keeping fixed lines
(number):     equidistant meshing with spacing of (number)
```

---

184 z mesh in units
Default *auto*

```
off:      Omit this direction for automatic meshing
auto:      automatic meshing
auto+fixed:     automatic meshing; keeping fixed lines
(number):     equidistant meshing with spacing of (number)
```

---

185 x mesh options in units
Default *off*

Define global mesh hint in x-direction

```
Values entered in units

min:      target minimum spacing
max:      target maximum spacing
force_min:     avoid spacing smaller than entered value
force_max:     avoid spacing larger than entered value
```

---

186 y mesh options in units
Default *off*

Define global mesh hint in y-direction

```
Values entered in units

min:      target minimum spacing
max:      target maximum spacing
force_min:     avoid spacing smaller than entered value
force_max:     avoid spacing larger than entered value
```

---

187 z mesh options in units
Default *off*

Define global mesh hint in z-direction

```
Values entered in units

min:      target minimum spacing
max:      target maximum spacing
force_min:     avoid spacing smaller than entered value
force_max:     avoid spacing larger than entered value
```

35.4. Simulation Box

---

188 User Defined Simulation Box
Default *Off*

```
Can be enabled if the values of the simulation box should be
set by the user.
```

35.4.1. Simulation Box/X

---

189 min
Default *0*

```
Set boundary co-ordinate at xmin:
- from structure: derive outermost co-ordinate from structure
- (number): Enter/Use value for boundary co-ordinate
```

---

190 max
Default *0*

```
Set boundary co-ordinate at xmax:
- from structure: derive outermost co-ordinate from structure
- (number): Enter/Use value for boundary co-ordinate
```

35.4.2. Simulation Box/Y

---

191 min
Default *0*

```
Set boundary co-ordinate at ymin:
- from structure: derive outermost co-ordinate from structure
- (number): Enter/Use value for boundary co-ordinate
```

---

192 max
Default *0*

```
Set boundary co-ordinate at ymax:
- from structure: derive outermost co-ordinate from structure
- (number): Enter/Use value for boundary co-ordinate
```

35.4.3. Simulation Box/Z

---

193 zmin
Default *0*

```
Set boundary co-ordinate at zmin:
- from structure: derive outermost co-ordinate from structure
- (number): Enter/Use value for boundary co-ordinate
```

---

194 zmax
Default *0*

```
Set boundary co-ordinate at zmax:
- from structure: derive outermost co-ordinate from structure
- (number): Enter/Use value for boundary co-ordinate
```

36. Simulation Options

36.1. Accuracy

---

195 Energy Decay
Default *auto*

Determine the Energy decay to stop the simulation

```
Energy decay in dB.

If set to "auto" this parameter is determined from the value
set in General - Structure Type.
```

---

196 Field Convergence Check
Default *off*

Field Monitor Convergence Check

```
This option acts as an additional end criterion and complements the
Energy Decay criterion.

The field which is recorded by field monitors is tested on convergence.
The simulation may only be stopped if the deviation is below the limit.
```

---

197 PGA
Default *on*

Perfect Geometry Approximation

```
Special Algorithm for rounded, off-grid and flat-metal structures.
Reduces staircase approximation error. Increases compilation
time but speed performance is maintained.

Note:
For testing purposes the PGA can be restricted to 2D or 3D objects
```

---

198 AC only simulation
Default *off*

Use Target frequency as excitation center

```
Using AC only excitation may yield better convergence,
if DC behaviour is not of interest.
```

36.2. Additional Space

---

199 xmin
Default *automatic*

Specify additional space at boundary xmin

```
Space can be entered in

- units (no suffix)
- wavelengthes of end frequency (suffix lambda)
- boundary spacing in % (suffix %)
```

---

200 xmax
Default *automatic*

Specify additional space at boundary xmax

```
Space can be entered in

- units (no suffix)
- wavelengthes of end frequency (suffix lambda)
- boundary spacing in % (suffix %)
```

---

201 ymin
Default *automatic*

Specify additional space at boundary ymin

```
Space can be entered in

- units (no suffix)
- wavelengthes of end frequency (suffix lambda)
- boundary spacing in % (suffix %)
```

---

202 ymax
Default *automatic*

Specify additional space at boundary ymax

```
Space can be entered in

- units (no suffix)
- wavelengthes of end frequency (suffix lambda)
- boundary spacing in % (suffix %)
```

---

203 zmin
Default *automatic*

Specify additional space at boundary zmin

```
Space can be entered in

- units (no suffix)
- wavelengthes of end frequency (suffix lambda)
- boundary spacing in % (suffix %)
```

---

204 zmax
Default *automatic*

Specify additional space at boundary zmax

```
Space can be entered in

- units (no suffix)
- wavelengthes of end frequency (suffix lambda)
- boundary spacing in % (suffix %)
```

36.3. Advanced

36.3.1. Advanced/Gabriel Object Options

---

205 Frequencies
Default *900e6*

```
List of frequencies for gabriel Objects.
```

---

206 Permittivity
Default *on*

```
Gabriel Objects yield permittivity and conductivity.
Switch 'off' to ignore the permittivity.
This may be useful and time-saving for low frequency
human body simulations up to approx. 1 MHz.
```

36.4. Boundary Conditions

---

207 xmin
Default *Absorbing 6 (> 40 dB)*

```
Electric    Etan=0 at boundary ( dark red line)
Magnetic    Htan=0 at boundary (green line)
Absorbing N    N=number of perfectly match layers (PML) (dotted line)
Absorbing Sheet    377 Ohm Sheet (dotted line)

Note:
The number in brackets give an estimation of the reflection
from the wall for one test case.
```

---

208 xmax
Default *Absorbing 6 (> 40 dB)*

```
Electric    Etan=0 at boundary ( dark red line)
Magnetic    Htan=0 at boundary (green line)
Absorbing N    N=number of perfectly match layers (PML) (dotted line)
Absorbing Sheet    377 Ohm Sheet (dotted line)

Note:
The number in brackets give an estimation of the reflection
from the wall for one test case.
```

---

209 ymin
Default *Absorbing 6 (> 40 dB)*

```
Electric    Etan=0 at boundary ( dark red line)
Magnetic    Htan=0 at boundary (green line)
Absorbing N    N=number of perfectly match layers (PML) (dotted line)
Absorbing Sheet    377 Ohm Sheet (dotted line)

Note:
The number in brackets give an estimation of the reflection
from the wall for one test case.
```

---

210 ymax
Default *Absorbing 6 (> 40 dB)*

```
Electric    Etan=0 at boundary ( dark red line)
Magnetic    Htan=0 at boundary (green line)
Absorbing N    N=number of perfectly match layers (PML) (dotted line)
Absorbing Sheet    377 Ohm Sheet (dotted line)

Note:
The number in brackets give an estimation of the reflection
from the wall for one test case.
```

---

211 zmin
Default *Absorbing 6 (> 40 dB)*

```
Electric    Etan=0 at boundary ( dark red line)
Magnetic    Htan=0 at boundary (green line)
Absorbing N    N=number of perfectly match layers (PML) (dotted line)
Absorbing Sheet    377 Ohm Sheet (dotted line)

Note:
The number in brackets give an estimation of the reflection
from the wall for one test case.
```

---

212 zmax
Default *Absorbing 6 (> 40 dB)*

```
Electric    Etan=0 at boundary ( dark red line)
Magnetic    Htan=0 at boundary (green line)
Absorbing N    N=number of perfectly match layers (PML) (dotted line)
Absorbing Sheet    377 Ohm Sheet (dotted line)

Note:
The number in brackets give an estimation of the reflection
from the wall for one test case.
```

36.5. Frequency

---

213 Start Frequency
Default *0 Hz*

Set range of frequency domain plot results

```
This parameter effects on DFT (discrete Fourier Transformation)
postprocessing and can be modified after simulation
```

---

214 End Frequency
Default *10 GHz*

Set range of frequency domain plot results

```
This parameter effects on DFT (discrete Fourier Transformation)
postprocessing and can be modified after simulation

The End Frequency is also a measure for the
grid accuracy used for the automatic meshing.
```

---

215 Number for Frequency Points
Default *200*

Set number of frequency domain points

```
This parameter effects on DFT (discrete Fourier Transformation)
postprocessing and can be modified after simulation
```

---

216 Target Frequency
Default *10 GHz*

For narrow band loss calculation, as
Default frequency etc.

```
The target frequency is used for:

- \newline Default frequency in Field Monitors
- Frequency in narrow band loss models
- Center frequency in AC-only simulations
```

36.6. General

---

217 Drawing Unit
Default *1 um*

One unit in drawing equals value in reality

```
1 unit in drawing can be set to
1e-9  = 1 nm
1e-6  = 1 um
1e-3  = 1 mm
1             = 1 m
25.4e-6  = 1 mil
25.4e-3  = 1 inch
```

---

218 Solvers
Default *EM*

Specify Solver for Simulation

```
EM:            FDTD solver
EM+Thermal:    FDTD solver + static FD-SOR
Thermal:       static FD-SOR
```

---

219 Structure Type
Default *SHR*

Set end criteria depending on structure type

```
This item effects on the following parameters:

- Maximum number of time steps (End criterion 1)
- Maximum Energy decay (End criterion 2)
- Resonance estimation order  (DFT postprocessing plot results)

These parameters set manually if User Setup is chosen.
```

---

220 Mesh Resolution
Default *Medium (15/4)*

Mesh Accuracy (Cells/wavelength,Cells/object)

```
This parameter effects on the mesh density determined by
- the minimum number of cells used for smallest wavelength or
- the minimum number of cells used for objects or gaps.

It also defines the number of search points for detecting small
objects in the simulation domain.

The resolution may also be specified numerically in side menu:
Automesh Setup - Advanced - Options - Mesh Resolution (Numeric)
with the parameter $NR, e.g. for automatic mesh refinement.

CW = Cells/Wavelength
CO = Cells/Object
SP = Search Points
NR = Numeric Resolution

Resolution,   CW, CO, SP , NR
-------------------------------------------
Coarse        10, 3,  5000,  1
Medium        15, 4, 10000,  5
Fine          20, 5, 20000, 10
Very fine     25, 6, 40000, 15
....
```

36.7. Loss Calculation

---

221 Dielectrics
Default *lossless*

Loss model for basic objects

```
Dielectric with a specified dissipation factor (loss tangent) can be
treated either lossless or lossy using a loss model, e.g. Debye.
The order and accuracy of the model is determined by the
bandwidth.

Special algorithms are applied which have impact on
the simulation speed. The loss calculation frequency band
is centered to the frequency range.

Note:
A conductivity specified for the dielectric is not affected by
this setting.
```

---

222 Conductors
Default *lossless*

Metal treatment for basic objects

Conductors can be treated either lossless or lossy
for a selectable frequency band.

Special algorithms are applied which have impact on
the simulation speed. The loss calculation frequency band
is centered to the target frequency.

Note:
- narrow, medium, wide band lossy models are thin sheet models
which account for the skin effect
- bulk loss model is recommended if sheet thickness is less than
skin depth

36.8. Meshing1

---

223 Mode
Default *Planar + 3D*

Mode selection

Planar	Mesh is optimized for flat metalization
Planar + 3D	Mixed mode
3D	Mesh is optimized for structures w/o flat metalization
User defined	Automesh parameters can be set by the user
Manual	No Automatic Meshing, automesh can still be triggered by the user.

---

224 Resolve Metal sheets
Default *thin*

Grid resolution of flat metal

If flat metals are detected this option determines if they are resolved by one
or two grid lines.

36.9. Meshing2

---

225 Algorithm
Default *Object Scan*

Meshing Algorithm

Algorithm for automeshing

- Object scan, fast, recommended
- Raytrace, slow, resolves priorities

36.10. Port Setup

---

226 Simulation Mode
Default *Separate Ports*

Defines Port Excitation method

```
Standard + Touchstone:
----------------------------------
Perform accurate multiport simulation and generate Touchstone file.
Tochstone file generation requires one simulation per port.
In case of a two or more port, this generates more accurate
S-Parameters than one-port excitations because the port load
impedance cancels out.

Seperate Ports:
-----------------------
Sequential Simulation with alternate port excitations

Active Impedance:
-------------------------
All ports excited simultaneously
```

36.11. Thermal

---

227 Ambient Temperature
Default *300*

Temperature of surrounding air

```
Used for convection, radiation and boundary heat sink definitions
```

---

228 Accuracy
Default *1e-2*

max. total error Power

```
End criterion for matrix solver.
Value should be much smaller than thermal source power.
```

---

229 Maximum Temperature
Default *5000*

Max. Temperature for divergence check

```
Iteration is truncated if this limit is achieved
```

---

230 Temperature Storage
Default *8 bit*

How will the temperature-field be stored ?

```
Number accuracy
```

---

231 Sides on Ambient Temperature
Default *None*

Sides of the bounding box to be on T_amb.

```
Acts as a heat sink with zero resistance,
e.g. ground plane
```

36.11.1. Thermal/SAR/ACD

---

232 Blood Density kg/m**3
Default *automatic*

Effects on tissue blood perfusion

```
Parameter rho_bl used in perfusion rate equation:

Qperf = rho_bl * C_bl * W_bl * rho_tiss * (T-T_bl)
```

---

233 Blood Heat Capacity J/kg/K
Default *automatic*

Effects on tissue blood perfusion

```
Parameter C_bl used in perfusion rate equation:

Qperf = rho_bl * C_bl * W_bl * rho_tiss * (T-T_bl)
```

---

234 Blood Temperature in K
Default *automatic*

Effects on tissue blood perfusion

```
Parameter T_bl used in perfusion rate equation:

Qperf = rho_bl * C_bl * W_bl * rho_tiss * (T-T_bl)
```

36.11.2. Thermal/Surface

---

235 Convection
Default *automatic*

Determination of surfaces for convection

Automatic:
Interfaces between objects and air will be determined automatically

Manual:
Convection area property defined by user

Off:
Convection is disregarded

---

236 Radiation
Default *automatic*

Determination of surfaces for radiation
Automatic:
Interfaces between objects and air will be determined automatically

Manual:
Radiation area property defined by user

Off:
Radiatin is disregarded

Part V.

Error Messages

This part gives a summery of error and warning messages and a comment to solve the problem.

37. GUI Messages

The following messages can appear during structure setup.

37.1. General

0100 "Rename Results: no results available"
At least one result folder sub-n has to exist

0101 "Another Simulation is running"
Use the batch mode for job queuing or a remote host

0102 "Invalid data passed to Expr()"
Please check the data format

0103 "Python Expression Check www.python.org"
Invalid syntax, please check the python expression

0106 "The Library Element expects POINTS on the Stack."
Please enter a valid point for operation

0107 "Default Arrow for a layer must be 1D."
Only arrows in x,y,z direction are valid

0108 "Syntax Error in Term"
Please check the Term for correct spelling

0112 "Could not open file 'patch.bat' for writing."
Update could not be completed, please check write permissions and retry

0113 "Save does not work without License"
Please install a valid license for this operation

0116 "Export does not work without License"
Please install a valid license for this operation

0117 "No excitation found in structure."
Simulation without an excitation is not possible. Please add a port or plane wave.

37.2. Fields and Animation

0200 "SAR and ACD only works if mass density != 0 specified in FDTD Elements"
Specify mass density in the material definition

0201 "Poynting Vector not avialable for time domain dumps"
Only possible for complex values (frequency domain)

0202 "No Electric field in dumpbox"
Electric field has not been stored in field dump

0203 "No Electric field in dumpbox needed for ACD/SAR"
ACD/SAR require storage of electric field

0204 "ieee/C95.3 or ieee/P1528 method selected for ACD calculation please select full or partial cube interpolation"

0205 "No Magnetic field in dumpbox"
Magnetic field has not been stored in field dump. Dumpbox should enclose at least 2 grid lines in each direction

0206 "No Magnetic field in dumpbox: needed for current calculation"
Magnetic field has not been stored in field dump. Dumpbox should enclose at least 2 grid lines in each direction

0207 "Dumpbox too thin for current calculation"
Magnetic field has not been stored in field dump. Dumpbox should enclose at least 2 grid lines in each direction

0208 "No Electric field in dumpbox needed for poynting vector calculation"
Electric field has not been stored in field dump

0209 "No Magneticic field in dumpbox needed for poynting vector calculation"
Magnetic field has not been stored in field dump. Dumpbox should enclose at least 2 grid lines in each direction

0210 "dumpbox has to be at least 2 cells size in all 3 dimensions for poynting vector calculation"
Magnetic field has not been stored in field dump.

0211 "Illegal animation plane for currents. Please increase the thickness of the animation Box"
Dumpbox should enclose at least 2 grid lines in each direction

37.3. Objects

0409 "MERGE: Objects with more than one text are forbidden."
Merging is only possible with objects having identical properties

0411 "MERGE: Objects with incompatible text."
Merging is only possible with objects having identical properties

0412 "Oversize only in plane views."
Switch to correct view for this operation

0413 "BOX to POLY conversion only in 2D views"
Switch to correct view for this operation

0414 "BOX to POLY conversion: Boxes are not uniform in perpendicular direction"
Operation only possible with equal-height boxes

0415 "Combine: Objects with more than one text are forbidden."
Combining is only possible with objects having identical properties

0416 "Combine: Objects with incompatible text."
Combining is only possible with objects having identical properties

0417 "Center impossible with MULTIBOXES."
Operation is only possible for single boxes

0418 "Legend only supported with 2D ARROWS"
Create 2D arrow for this operation (no perpendicular coordinate)

0419 "Mirroring only supported with 1D ARROWS"
Create 1D arrow, directed in either x,y, or z direction

0420 "Rotation supported for 2D Points only"
Create point in current view

0422 "No Handles Selected"
Operation requires active (green) handles, pick with left mouse button

0423 "A range of neighbor Elements has to be selected for this operatrion"
Select handles, pick with left mouse button

0426 "Path to Poly only in plane views."
Switch to polygon plane for this operation

0427 "Path to POLYS with arrows perpendicular to view plane only."
Arrows have to be perpendicular to the polygon plane

0430 "Only 2D Objects are permittet in POLYs."
Enter 2D points in the polygon plane for this operation

0431 "Only 2D Objects PERPENDICULAR to the current layer's default ARROW are permittet in POLYs."
Enter 2D points in the polygon plane for this operation

0432 "Exactly two connected POLY Elements have to be selected for this operation."
Select exactly 2 subsequent handles of the polygon

0434 "Improper Selection for Polygon Bisection"
Select exactly 2 non-neighboring handles

0435 "Object Height must be 1D."
Only 1D arrows are allowed for height definition

0436 "Height cannot be assigned to Object."
Object has to be a polygon or a box

0438 "Generate ROTPOLY with 1D Axis only"
Arrow has to be directed in either x, y, or z direction

0439 "ROTPOLY Axis must be in the POLY plane."
Arrow has to be perpendicular to the polygon plane

0441 "Set Tangential: 2D viewport only."
Select the right view for this operation

0442 "LINPOLY: Begin of line is not in grid region"
Enlarge simulation box

0443 "LINPOLY: Line is leaving grid region"
Enlarge simulation box

0444 "Exactly two elements have to be selected for length measurement"
Select exactly 2 polygon handles for this operation

0449 "No Objects/Layers for Autodisc"
Automatic meshing requires objects or ports

0450 "Only 2D Objects are permittet in Bond Wire creation."
Enter 2D points in the polygon plane for this operation

0451 "Only 2D Objects are permittet in CYLINDER creation."
Enter 2D points in the polygon plane for this operation

0452 "Subdivide ¿ 500"
Decrease the number for subdivision

0454 "Symmetry check with 1d Arrows only"
Only arrows directed in either x, y, oz z direction are allowed for this operation

0455 "Discretization not possible with less than 2 discretization lines in direction."
Check meshing in the respective direction

0457 "Rotation of discretization in wrong view."
Select the correct view for this operation

0458 "Rotation of discretization for +/- 90 Degrees only"
Please check the input number for rotation angle

0459 "FLIP: only 90 and -90 degrees permitted"
Please check the input number for rotation angle

0461 "Not enough objects for poly creation"
At least 2 points are required for polygon creation

0463 "Perpendicular Arrow (current layer's default ARROW) for a POLY must be 1D."
Only arrows directed in either x, y, oz z direction are allowed for this operation

0466 "Illegal Object in POLY creation:"
Only 2D points are allowed in polygon creation

0468 "Multiple physical properties are not permitted"
Check properties of respective layer or object

0469 "Object with incompatible multiple texts"
Check properties of respective layer or object

0470 "SMD Type unknown"
Check SMD definition

0471 "Attempt to save discretized structure"
Please use undo to recover original structure

0474 "Objects with incompatible heights selected."
Operation can only be perfromed with equal height objects

0475 "Sorry, but a Material with the name already exists"
Choose another name for this entry

0476 "Sorry, no name chosen."
Enter a name

0477 "Operand is missing for Operation"
Enter the missing operand

37.4. Import and Export

0600 "export_3d: only possible in 3D view mode"
Switch to 3D Tab for this operation

0601 "Please Save Structure before Gerber Export"
Alternatively, use 'Save as' if saving of current structure is not wanted

0602 "GDSII-Export: Polygon with ¿ 8190 Points exceeds GDSII specification on layer"
Use 'simplify polygon' or 'bisect polygon' to reduce number of points

0603 "Wrong Type"

0604 "All Layers must have default arrows with the same direction"
PLease check the layer heights

0605 "Malformed circle aperture definition"
Aperture can not be created, check import data

0606 "No Y dimension in rectangle aperture definition"
Aperture can not be created, check import data

0607 "Malformed rectangle aperture definition"
Aperture can not be created, check import data

0608 "No Y dimension in oval aperture definition"
Aperture can not be created, check import data

0609 "Malformed oval aperture definition"
Aperture can not be created, check import data

0610 "Malformed aperture definition for regular polygon"
Aperture can not be created, check import data

0611 "Malformed polygon aperture definition"
Aperture can not be created, check import data

38. Compilation Messages

The following messages can appear during pre- or post processing and are displayed in the log window.

2100 "no discretisation lines in at least one direction"
Check the mesh in each direction

2102 "Unexpected End of ACAD File"
The file could be corrupted. Please try 'Convert & Save' operation for recreation

2103 "No Host Enabled"
Please select a host in the host list

2105 "Process exit with error."
Check the log window or log files for error message

2106 "The structure contains no Farfield-box.Try the refresh button."
Update the simulation control or check the far field property definition

2107 "The structure contains no Cached SAR/ACD boxes."
This option was not enabled before simulation. Check the anmiation property definition.

2108 "No Optimisation or Batch to start"
Please add a batch, parameter sweep or optimization first

2109 "Only S-Parameter Files Supported"

2110 "Unknown Format"
Only files with known format are supported

2111 "Incompatible Files"
Files in equations have to be of the same format and size

2112 "A Complex Number is required for multiplication"
Please enter a complex number, e.g. complex(1,0)

2113 "Incompatible Fields for superposition"
Files in equations have to be of the same format and size

2114 "Error: trying to combine incompatible files."
Files in equations have to be of the same format and size

2115 "Syntax error in Source Term"
Please check the equation for correct syntax

2117 "TOUCHSTONE GENERATION: incompatible filelengths"
Source files have different filelength, try new postprocessing

2118 "TOUCHSTONE GENERATION: incompatible frequencies"
Source files have different frequencies, try new postprocessing

2119 "SPICE Calculation: no frequencies found"
Check frequency range for SPICE netlist generation

2120 "Can not make directory Path name too long or wrong permissions ?"
Please check write permissions or shorten path name

2121 "Optimization Goal: File does not exist"
Please check the existence of the source file(s)

2122 "Illegal Name for SOURCE term"
Please check the source term in goal definition

2123 "Incompatible SOURCES in one TERM in goal"
Please check the source terms in goal definition

2124 "No SOURCE used in TERM in goal"
Please define a source term the goal definition

2125 "Reference is zero in goal"
Please check the goal definition

2126 "No Parameter point (frequency/time/angle) meets restrictions in goal. Parameters were restricted from by the user."
Please check the goal definition

2129 "License Checkout for Optimizer failed."
Please install a valid license

39. Kernel Messages

The following messages can appear during simulation run time and displayed in the log window.

3000 "Syntax Error in Property: –¿ Parameter Expected.Property is"
Please check the property definition

3001 "Port meshing problem, please check port resolution (voltage)."
Please check the mesh lines in the area of the port

3002 "Current Box has to be in between lines in exactly one direction."
Please check the mesh lines in the area of the port

3003 "Delay ¡ 0 not allowed"
Please check the delay definiton of the exciting port

3004 "Delay ¿ 1 second found."
Please check the delay definiton of the exciting port

3005 "Diode too short, NRS ¡= length"
Increae the lenght of the diode

3007 "Excite boxes must be at exactly one boundary"
Absorbing ports have to located at the boundaries, check simulation box

3008 "Incompatible multiple excite boxes"
Please check the port definition

3009 "WG Port has to be flat in exactly one direction."
The waveguide port cross section must cover the outermost grid line, only

3015 "PATH field computation for frequency domain dumps only"
Field dump contains time domain data

3016 "Zero Path length"
Either only one point defined or two points are too close to each other

3018 "ERROR: double dumpbox filename"
Please check the input file for double field dumps

3019 "Excitation box (current) must not be flat in perpendicular directions"
Please check the mesh lines in the area of the port

3020 "Excitation box (voltage) must not be flat in excitation direction"
Please check the mesh lines in the area of the port

3024 "Multiple Mirror Planes in direction"
Only one mirror plane can be applied in far field calculation

3025 "No proper plane_wave setup for backscattering."
Please check the plane wave definition

3026 "Empty Structure: All cells are metallic, no field"
Please check the input file for wrong conducting volumes

3031 "Not enough hosts enabled for Multi-PC with hosts."
Please check the Multi-PC set up and select sufficient hosts

3032 "MPC: connection failiure."
Please check the availability of all hosts

3033 "Illegal voltage path for"
Please check the mesh lines in the area of the port

3034 "Illegal current path for"
Please check the mesh lines in the area of the port

3038 "conductor / dielectric: illegal syntax:"
Please check the property definition

3039 "skin: all skin objects must have same tmax and same accu and same sigma"
Please check the property definition

3040 "LICENSE CHECKOUT FAILED FOR"
Please install a valid license

3042 "Waveguide is fully filled (e.g. magnetic boundary, lossy dielectrics)"
Please check the waveguide cross section

3043 "Waveguide is fully filled (e.g. electric boundary, lossy dielectrics)"
Please check the waveguide cross section

3044 "Waveguide with inhomogenious dielectric Filling"
Please check the waveguide cross section

3045 "ERROR: No PML boundary condition at waveguide port"
Please check the waveguide boundary

3046 "Not Enough Memory for On-Disc/Multi PC Calculation of this structure on node"
Please select a suitable host

Part VI.

Theoretical Background

40. General Description

The EMPIRE XPU™ simulator is a tool for solving Maxwell's equations, especially for radio frequency (RF) applications. It is based on the Finite Difference Time Domain method (FDTD), which means that the equations are discretized in space and time. This is accomplished by mapping the structure of interest onto a rectangular mesh where the unknown field components are located in each cell.

Because of the nature of the electromagnetic problem[1], an initial value problem has to be solved. This means that the unknown field for a certain time is calculated from the field values one timestep before. The FDTD method employs an efficient time stepping algorithm, known as the Yee's leapfrog scheme [3] to solve the initial value problem. The size of the time steps is related to the size of the smallest cells for stability reasons and can not, therefore, be defined independently. So the definition of a suitable mesh is an important task for efficient simulation.

40.1. Simulation Time

The duration of a simulation run is proportional to the number of cells and time steps needed to reach steady state.

$$T_{sim} = \frac{N_{cells} N_{timesteps}}{Performance} \tag{40.1}$$

40.2. Performance

The performance depends on the

- hardware (number of cores, cache size, number of CPUs, ...)
- software (XPU algorithm, cluster algorithm)
- structure (boundary conditions, loss models, field dumps, ...)
- excitation (Concentrated source, matched source)

The peak performance can be checked in EMPIRE XPU™ by running a **Kernel Speed Test** in the **Utilities** menu.

[1] Maxwell's equations can be reformulated as hyperbolic partial differential equations

40.3. Number of Timesteps

The required number of timesteps depends on the quality factor of the system, which can not be determined easily. For a low Q system the number can be estimated by the pulse length and propagation time needed:

$$N_{timesteps} = \frac{T_{pulse} + \frac{D}{c}}{\delta t} \tag{40.2}$$

Here, D is the diameter of the simulation volume or line length, c is the velocity of light and δt is the time step size 43.2.

40.4. Number of Cells

The number of cells depend on the structure resolution and the maximum frequency which require a minimum sampling s per minimum wavelength. If the structure details can be neglected this number can be estimated to

$$N_{cells} = (\frac{s f_{max}}{c})^3 D_x D_y D_z \tag{40.3}$$

Where D_x, D_y, D_z are the extensions of the simulation volume.

Due to the limited memory resource of a computer, only a finite number of cells can be processed. EMPIRE XPU™ does a very efficient memory allocation by re-using coefficients and on-the-fly code generation. Without field dumps and boundary conditions the memory usage can be estimated to

40.5. Simulation Memory

$$Memory = N_{cells} * 24 Bytes \tag{40.4}$$

The simulation area is terminated with boundary conditions which will be explained in Chapter 42.

In certain cases, instabilities can occur before reaching the steady state. The reasons for this and some rules to overcome this kind of problems are explained in Chapter 43.

40.6. Simulation Assumptions

The aim of the FDTD method is to solve Maxwell's equations and its associated material relations for a given structure and boundary conditions.

$$\nabla \times \vec{H} = \dot{\vec{D}} + \vec{J}, \tag{40.5}$$

$$\nabla \times \vec{E} = -\dot{\vec{B}}, \tag{40.6}$$

$$\vec{D} = \varepsilon \vec{E}, \tag{40.7}$$

$$\vec{B} = \mu \vec{H}, \tag{40.8}$$

$$\vec{J} = \sigma \vec{E}, \tag{40.9}$$

where $\vec{E}$ and $\vec{H}$ are the electric and magnetic field vectors, $\vec{D}$ and $\vec{B}$ are the electric and magnetic flux density vectors and ε, μ, and σ are permittivity, permeability, and conductivity of the material, respectively. Some assumptions have to be made to derive the algorithm to solve these equation with numerical methods:

- Passivity
  The structure does not contain any active element. These elements can not be described by Maxwell's equations and, therefore, have to be excluded from the simulation. A method to include those elements as lumped elements connected at inner ports using the so called compression approach will be explained in chapter 46.
- Linearity
  Non-linear elements, like semi-conductors and some dielectric or permeable materials, need special treatment in the algorithm, which is not considered here.
- No remedy effects
  Hysteresis effects, which can occur in highly permeable materials, are not considered in the simulator.
- Isotropy
  Anisotropic materials can be taken into account in the algorithm as long the material tensor can be described by a diagonal matrix which must coincide with the co-ordinate system of the grid. Because of higher memory requirements, this feature is not implemented yet.
- Absence of free charges
  Free charges can be accelerated by electromagnetic fields and, therefore, have to be calculated with a combined set of Maxwell's equations and movement equations, which is often implemented in so called Particle in Cell Codes.
- Frequency independent materials
  For the applied algorithm, the material properties are assumed to be independent of frequency, because often a whole frequency range is covered by exciting with Gaussian-like pulses. If the frequency dependence can be described by simple differential equations in the time domain it can be considered but the algorithm has to be changed for those regions (e.g. Debye, Gabriel, Plasma, Meta materials)

40.7. The Yee algorithm

No assumptions have to be made for the shape and material distribution of the structure. The only limitation is that the structure has to be subdivided into cells in which the material properties are constant. For the Finite Difference scheme, these cells are formed by intersecting planes of a Cartesian co-ordinate system. The basic idea of the algorithm is to place the unknown field components in a certain position of each cell so that every electrical field component $\vec{E}$ is surrounded by four circulating magnetic field components $\vec{H}$ and vice versa, as shown in Figure 40.1, and to approximate Maxwell's differential equations by central differences.

For example, the corresponding approximate equation for the $\dot{B}_\mathrm{x}$ component gives

$$\dot{B}_\mathrm{x} = \frac{E_\mathrm{y}(x,y,z+\frac{\delta z}{2}) - E_\mathrm{y}(x,y,z-\frac{\delta z}{2})}{\delta z} - \frac{E_\mathrm{z}(x,y+\frac{\delta y}{2},z) - E_\mathrm{z}(x,y-\frac{\delta y}{2},z)}{\delta y} \tag{40.10}$$

Also the time derivative is approximated by central differences and arranged in such a way, that the computed values for $\vec{B}$ and $\vec{E}$ are shifted for a half step in time, yielding the well-known leapfrog scheme. For any component A, a numbering according to

$$A(x,y,z,t) = A\left(\sum_i \delta x_i, \sum_j \delta y_j, \sum_k \delta z_k, n\delta t\right) \hat{=} A^n_{i,j,k} \tag{40.11}$$

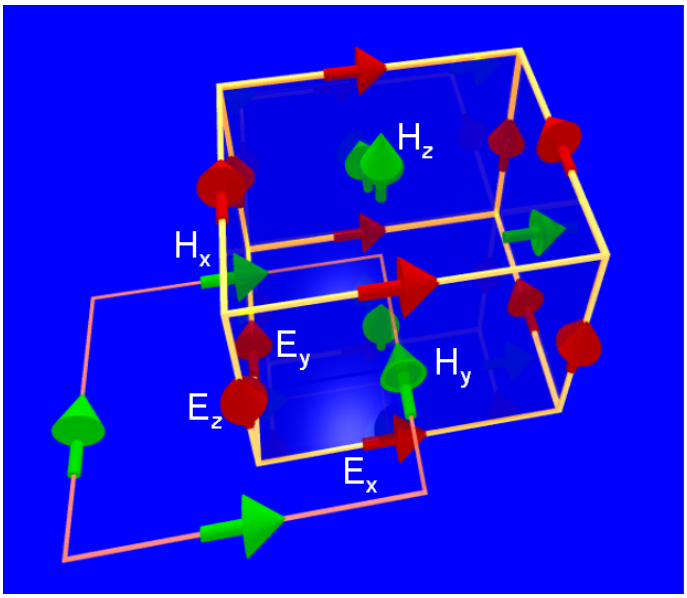


Figure 40.1.: Arrangement of field components in a Yee cell

is introduced. The FDTD scheme can then be written as:

$$E^{n+1}_{\mathrm{x}_{i^+,j,k}} = \frac{1-\frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}}{1+\frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}} E^{n}_{\mathrm{x}_{i^+,j,k}} + \frac{2\frac{\delta t}{\varepsilon_{ijk}}}{1+\frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}} \left(\frac{H^{n+1/2}_{\mathrm{z}_{i^+,j^+,k}} - H^{n+1/2}_{\mathrm{z}_{i^+,j^-,k}}}{\delta y_j + \delta y_{j-1}} - \frac{H^{n+1/2}_{\mathrm{y}_{i^+,j,k^+}} - H^{n+1/2}_{\mathrm{y}_{i^+,j,k^-}}}{\delta z_k + \delta z_{k-1}} \right) \tag{40.12}$$

$$E^{n+1}_{\mathrm{y}_{i,j^+,k}} = \frac{1-\frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}}{1+\frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}} E^{n}_{\mathrm{y}_{i,j^+,k}} + \frac{2\frac{\delta t}{\varepsilon_{ijk}}}{1+\frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}} \left(\frac{H^{n+1/2}_{\mathrm{x}_{i,j^+,k^+}} - H^{n+1/2}_{\mathrm{x}_{i,j^+,k^-}}}{\delta z_k + \delta z_{k-1}} - \frac{H^{n+1/2}_{\mathrm{z}_{i^+,j^+,k}} - H^{n+1/2}_{\mathrm{z}_{i^-,j^+,k}}}{\delta x_i + \delta x_{i-1}} \right) \tag{40.13}$$

$$E^{n+1}_{\mathrm{z}_{i,j,k^+}} = \frac{1-\frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}}{1+\frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}} E^{n}_{\mathrm{z}_{i,j,k^+}} + \frac{2\frac{\delta t}{\varepsilon_{ijk}}}{1+\frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}} \left(\frac{H^{n+1/2}_{\mathrm{y}_{i^+,j,k^+}} - H^{n+1/2}_{\mathrm{y}_{i^-,j,k^+}}}{\delta x_i + \delta x_{i-1}} - \frac{H^{n+1/2}_{\mathrm{x}_{i,j^+,k^+}} - H^{n+1/2}_{\mathrm{x}_{i,j^-,k^+}}}{\delta y_j + \delta y_{j-1}} \right) \tag{40.14}$$

$$H^{n+1/2}_{\mathrm{x}_{i,j^+,k^+}} = H^{n-1/2}_{\mathrm{x}_{i,j^+,k^+}} + \frac{\delta t}{\mu_{ijk}} \left(\frac{E^{n+1/2}_{\mathrm{y}_{i,j^+,k+1}} - E^{n+1/2}_{\mathrm{y}_{i,j^+,k}}}{\delta z_k} - \frac{E^{n+1/2}_{\mathrm{z}_{i,j+1,k^+}} - E^{n+1/2}_{\mathrm{z}_{i,j,k^+}}}{\delta y_j} \right) \tag{40.15}$$

$$H^{n+1/2}_{\mathrm{y}_{i^+,j,k^+}} = H^{n-1/2}_{\mathrm{y}_{i^+,j,k^+}} + \frac{\delta t}{\mu_{ijk}} \left(\frac{E^{n+1/2}_{\mathrm{z}_{i+1,j,k^+}} - E^{n+1/2}_{\mathrm{z}_{i,j,k^+}}}{\delta x_i} - \frac{E^{n+1/2}_{\mathrm{x}_{i^+,j,k+1}} - E^{n+1/2}_{\mathrm{x}_{i^+,j,k}}}{\delta z_k} \right) \tag{40.16}$$

$$H^{n+1/2}_{\mathrm{z}_{i^+,j^+,k}} = H^{n-1/2}_{\mathrm{z}_{i^+,j^+,k}} + \frac{\delta t}{\mu_{ijk}} \left(\frac{E^{n+1/2}_{\mathrm{x}_{i^+,j+1,k}} - E^{n+1/2}_{\mathrm{x}_{i^+,j,k}}}{\delta y_j} - \frac{E^{n+1/2}_{\mathrm{y}_{i+1,j^+,k}} - E^{n+1/2}_{\mathrm{y}_{i,j^+,k}}}{\delta x_i} \right) \tag{40.17}$$

Here, the superscripts $i^{\pm}$ denote a spatial shift of half a cell size. The discrete material parameters $\sigma_{ijk}, \varepsilon_{ijk}, \mu_{ijk}$ are obtained from the continuous ones by a special averaging scheme. They are stored in an efficient way, so that the memory requirement is approximately limited to the 6 components.

These equations can now be solved recursively if initial values $\vec{E}^0$ and $\vec{H}^{1/2}$ and proper boundary conditions are given. The field values are updated in each time step and lost, if they are not explicitly saved as done when recording voltages, currents or field plots. As can be seen, no matrix inversion is needed and so the

calculation with numbers having medium precision (4 Bytes) is sufficient for most applications, thus leading to a very low memory requirement (appr. 24 Bytes per cell) for the simulation.

41. Discretization

The structure to be analyzed has to be mapped on the FDTD mesh. Even if more general meshes could be constructed to be used for the Finite Difference scheme, the Cartesian grid[1] has proven to be the most efficient and versatile one and is applied in the EMPIRE XPU™ software.

41.1. Meshing

To maintain second order accuracy of the FDTD scheme, an equidistant spacing should be preferred. But to reduce memory requirements, it is sometimes advisable to change the spacing of the grid lines according to the expected change of field strength.

Hint

The spacing of the mesh lines should fulfill the following conditions.

- Sampling resolution SR: The space has to be resolved in such way, that the smallest wavelength is sampled properly
- Detail resolution DR: Strip width, dielectric sheets or gaps should be resolved with a few cells.

Both numbers for the resolution can be adjusted in EMPIRE XPU™ with the **Mesh resolution** which is $SR = 15$ and $DR = 4$ by default **Medium**.

Remarks:

- In the region of edges and corners the field values tend to high values. EMPIRE XPU™ features the "1/3rd" rule to account for metalization edges to reduce the simulation effort
- EMPIRE XPU™ supports thin sheet models for metalization, so skin depth does not need to be resolved
- For curved or off-axis geometry EMPIRE XPU™ features a local conformal meshing (PGA, Perfect Geometry Approximation)

Hint

It is recommended to preview the meshed structure of all objects. This can be done by using the function **Meshed 3D Structure**

[1] formed by intersecting planes of Cartesian co-ordinates

42. Boundary Conditions

The calculation area has to be truncated by applying boundary conditions at the outer mesh cells. Due to the specified condition, special algorithms have to be applied to the field components.

42.1. Electric Wall

Applying this condition will force the tangential electric field components at the outer plane to be zero. In other words, only normal electric field components exist, so the electric field is perpendicular to this wall. There are two types of application for using an electric wall:

- The boundary is covered by a perfect conductor
- The structure and the excitation have a symmetry plane on which the electric field is perpendicular.

42.2. Magnetic Wall

Applying this condition, it will force the tangential magnetic field components at the outer plane to be zero. In other words, only normal magnetic field components exist, so the magnetic field is perpendicular to this wall. The only application for this boundary condition is a symmetry plane where the magnetic field is perpendicular. Due to the locations of the tangential magnetic field components in the grid, the magnetic wall can only be placed in the middle of the first cell as illustrated in Figure 42.1.

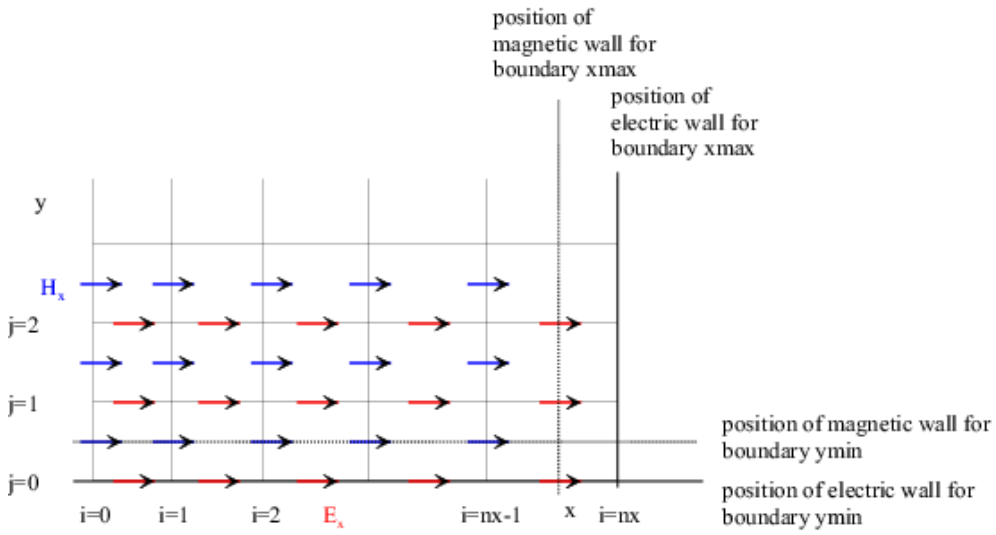


Figure 42.1.: Position of electric and magnetic walls in the grid

Hint

- The plane of symmetry for a magnetic wall lies between the two outermost grid lines of the mesh.
- The field values behind the magnetic wall can not be accessed.
- A weighting factor of two has to be defined within a current integration path definition, if a conductor is halved by a magnetic wall to give a proper impedance value.

42.3. Absorbing Boundaries

If outgoing waves should not be reflected into the simulation domain absorbing boundaries have to be applied. Those boundary condition cause additional computational effort and a compromise between simulation performance and absorption quality has to be found.

These boundaries should be placed apart from the radiating source to ensure that reactive fields do not reach the boundary considerably.

Therefore, EMPIRE XPU™ adds additional space at an Absorbing Boundary in the case of

- Definition of a far field monitor
- Structure Type set to **Antenna** or **Inductor**

42.3.1. Absorbing sheet

A simple absorbing boundary can be realized by a resistive sheet with the free space impedance of 377 Ohm. This boundary condition can be applied for highly directive radiation towards the boundary. It can also be used for non-radiating structures (e.g. circuits) to suppress box modes.

42.3.2. Perfectly Matched Layer

This boundary condition is well suited for absorbing outgoing waves having arbitrary directions of propagation. It is realized by employing gradual electric and magnetic conductivities at the outer walls. The quality of the absorption can be improved by enlarging the conducting thickness. The thickness of the conducting area can be chosen by specifying the number of layers in the parameter setup.

Hint

As a rule of thumb, the distance between a radiating source and an absorbing wall with `pml` boundary conditions should be at least a quarter of a wavelength.

Hint

The boundary employing `pml` boundary conditions may be inhomogeneous (e.g. layered dielectrics), but the material may not be lossy.

43. Stability Conditions

An undesired property of the explicit time integration scheme in the FDTD method is the possibility of an unlimited growth in time of the computed values. In general, this can be overcome by selecting a proper value for the time step δt.

43.1. Uniform grid

To ensure stability of the Yee-algorithm for a uniform grid with homogeneous material distribution, a proper time step can be derived from an eigenvalue analysis of the discrete differential operators in both space and time which yields

$$\delta t \leq \frac{1}{c_0 \sqrt{\frac{1}{(\delta x)^2} + \frac{1}{(\delta y)^2} + \frac{1}{(\delta z)^2}}}. \tag{43.1}$$

As can be seen, if the grid spacing is chosen very dense, the time step has to be very small which leads to long simulation runs. Equation (43.1) can also be applied to inhomogeneous regions, since it represents the worst case.

43.2. Non-uniform grid

If the grid spacing is varying an exact stability criterion can not be derived and, therefore, an empirical upper bound for the time step is applied as displayed in (43.2).

$$\delta t \leq \frac{1}{c_0 \max_{i,j,k} \left(\sqrt{\frac{1}{\mu_{x_{ijk}} \varepsilon_{x_{ijk}} (\delta x_i)^2} + \frac{1}{\mu_{y_{ijk}} \varepsilon_{y_{ijk}} (\delta y_j)^2} + \frac{1}{\mu_{z_{ijk}} \varepsilon_{z_{ijk}} (\delta z_k)^2}} \right)} \tag{43.2}$$

43.3. Open boundaries

As for the non-uniform grid, stability conditions can not be derived, especially for artificial boundaries, like absorbing boundary conditions. In fact, instabilities have been reported in [4] when applying open boundary conditions to layered dielectrics.

Some empirical rules can be given to avoid instabilities:

Hint

- equidistantly spaced grid lines before the boundary
- at least $\frac{\lambda}{4}$ distance from a source to a boundary
- avoid reactive fields at the boundaries by increasing the boundary distance
- use an Increased Stability Reserve within the parameter set-up

44. Excitations

For solving the initial value boundary problem, the excitation type and shape must be given for the whole simulation time.

44.1. Types of excitation

Although it might be possible to impress the initial field arbitrary, only physically reasonable excitations lead to the desired results.

44.1.1. Concentrated source excitation

This source is used for excitation inside the simulation volume. This source excites an electric field between two conductors and a load is placed in parallel for the termination. For feed-line ports a TEM-like propagation of the wave can be assumed after a certain length and voltages and currents can be recorded in the reference plane. In case of lumped ports, voltages and currents are recorded at the load location.

44.1.2. Matched source excitation

This excitation type assumes a TEM-like propagation of the wave at the boundary. So, the structure at the boundary is extended to a certain length $L_{\text{source}}(44.1)$ where at the beginning the field is impressed according to the excite definition. Naturally, the impressed field is not the physical mode, but as the wave propagates along the line the desired physical mode will be established in the port plane.

$$L_{\text{source}} = n_{\text{source}} b_{\text{dis}} \tag{44.1}$$

Here, the parameter n_{source} is entered in the parameter setup and b_{dis} is defined by the first grid spacing at the excitation plane.

Hint

As a rule of thumb, the length of the matched source should be more than twice the diameter of the effective field cross section.

44.1.3. Plane wave excitation

The excitation by plane waves is realized as a compact source, where outside the source only the scattered field exists as depicted in Figure 44.1.

Inside the source both incident waves and scattered fields are present. One electric boundary may be applied [1] when plane waves are used.

[1] to simulate a ground plane

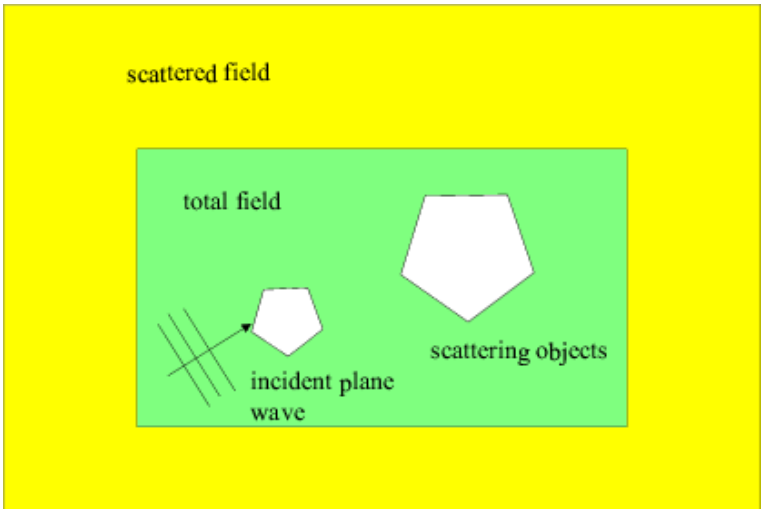


Figure 44.1.: Plane wave excitation

Hint

For plane wave excitation the scattering objects must lie inside the excitation box. Objects lying outside or intersect the excitation box will lead to unphysical results.

The plane wave may have arbitrary direction of propagation which is specified by the spherical co-ordinates θ and φ and arbitrary polarization, specified by E_θ and E_φ.

More than one plane wave can be excited for exploiting interference effects. With this feature, elliptical polarizations or standing waves can be simulated. By default, the shape of the excitation is the Gaussian pulse. The shape may be varied according to section 44.2.

To specify the desired values in spherical co-ordinates which are given in Cartesian co-ordinates, the following conversion formulas are useful.

$$\begin{pmatrix} E_\theta \\ E_\varphi \end{pmatrix} = \begin{pmatrix} \cos(\varphi)\cos(\theta) & \sin(\varphi)\cos(\theta) & -\sin(\theta) \\ -\sin(\varphi) & \cos(\varphi) & 0 \end{pmatrix} \cdot \begin{pmatrix} E_\mathrm{x} \\ E_\mathrm{y} \\ E_\mathrm{z} \end{pmatrix} \tag{44.2}$$

44.1.4. Waveguide excitation

Hollow waveguides can be excited by TE- or TM-modes which are pre-calculated by a two-dimensional eigenvalue solver. The cross-section may be arbitrary but must be enclosed by metal. The cross-section may be homogeneously filled by a dielectric material. The frequency range may be arbitrary but propagation of the exciting mode is only possible if the frequency is above the mode's cut-off. The dispersion relation 44.3 of a dielectric filled waveguide is known.

$$\beta = \frac{2\pi}{c_0}\sqrt{\varepsilon_r f_0^2 - f_c^2} \tag{44.3}$$

Here, f_c is the cut-off frequency of the hollow waveguide.

44.2. Time signals

By default, the time signal of the structure excitation is the Gaussian pulse where the modulation frequency f_0 and the bandwidth $2f_c$ is defined by the user.

$$\mathrm{exc}(t) = \cos\left[2\pi f_0 (t - t_0)\right] e^{-\left(\frac{t-t_0}{\tau}\right)^2} \tag{44.4}$$

This definition leads to the following properties of the pulse:

- The amplitudes in frequency domain at the borders $f_0 \pm f_c$ are -20dB of the amplitude of the center frequency f_0.
- The amplitude in time domain at the first step $t = 0$ is -100dB.
- The time duration for the incident pulse is $2t_0$.

The time signal can also be selected from a list of templates or edited by the user. Many mathematical function are supported. The excitation signal can be previewed in the window and the corresponding DFT is shown immediately.

In order to save memory the time signals are stored not every time step but a sample factor is used:

$$NS = int(\frac{1}{f_{max} * DT * 2 * N_{over}}) \tag{44.5}$$

where f_{max} is the maximum frequency, DT is the timestep and N_{over} is the oversampling factor which can be set in the **Excitation & End Criteria** section.

45. Ports and S-matrices

Some important results when analyzing RF elements with a field simulator are scattering parameters, representing reflected and transmitted waves at the ports. When the structure is closed and the ports are defined unique, the whole passive structure can be described by a frequency dependent scattering matrix as depicted in Figure 45.1.

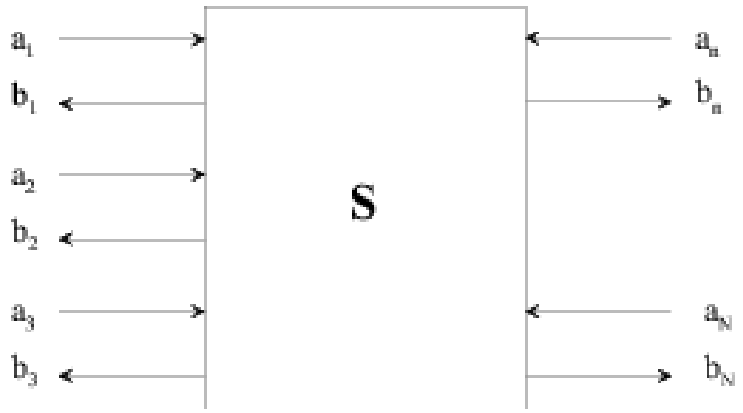


Figure 45.1.: Multiple port scattering matrix

The elements of the scattering matrix are obtained by exciting the structure N times with the wave quantities $a_n, n = 1 \ldots N$, under the condition that all ports are terminated by a matched load.

$$s_{i,j} = \left.\frac{b_i}{a_j}\right|_{a_k=0 \,\forall\, k \neq j} \tag{45.1}$$

If radiation can occur, as it does for open structures, the leakage of radiated power can be evaluated applying the law of energy conservation.

$$P_{\text{loss}} = \frac{1}{2}\sum_{n=1}^{N} |a_n|^2 - |b_n|^2 \tag{45.2}$$

45.1. Port definition

A port can be defined as a plane, where incident and reflected waves can be represented by complex wave quantities a and b, respectively.

45.1.1. Quasi-TEM ports

The most common ports for analyzing passive structures are single mode waveguides[1] where voltages and currents can be uniquely defined. Then the wave quantities can be obtained by

$$a_n = \frac{u_n^{\mathrm{inc}}(f)}{\sqrt{Z_n}} = i_n^{\mathrm{inc}}(f)\sqrt{Z_n} \tag{45.3}$$

$$b_n = \frac{u_n^{\mathrm{ref}}(f)}{\sqrt{Z_n}} = i_n^{\mathrm{ref}}(f)\sqrt{Z_n} \tag{45.4}$$

$$Z_n = \frac{u_n^{\mathrm{inc}}(f)}{i_n^{\mathrm{inc}}(f)} = \frac{u_n^{\mathrm{ref}}(f)}{i_n^{\mathrm{ref}}(f)} \tag{45.5}$$

Here, Z_n is the characteristic impedance of the line at port n. To define a port, both incident and reflected voltages and currents have to be sampled during the simulation. Here, EMPIRE XPU™ features source excitation where the mode is excited correctly over the whole frequency range. The source delivers the voltage and current in the reference plane. In Figure 45.2, a microstrip port is depicted as an example. The voltage

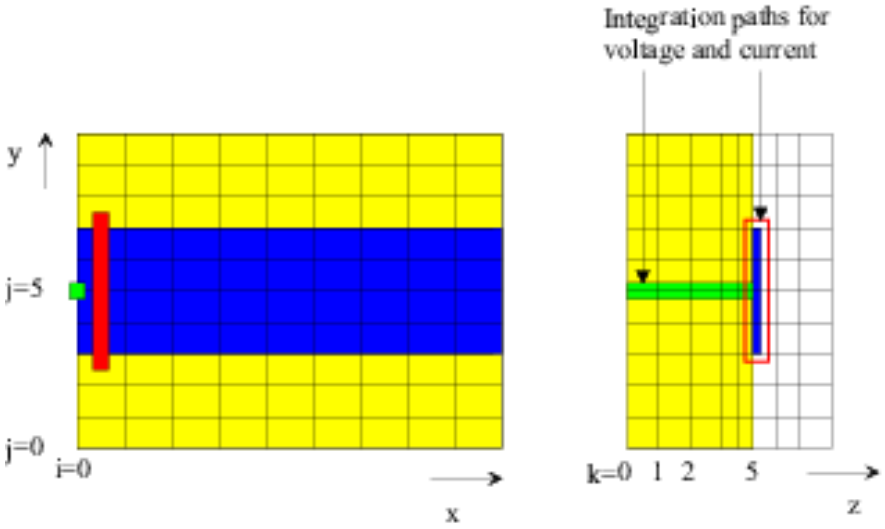


Figure 45.2.: Illustration of an outer microstrip port

is obtained by integrating numerically the corresponding electrical field vectors located on the grid lines along the path specified by the box at $i = 0$ and $j = 5$.

$$u(t) = \int_0^h E_z \mathrm{d}z \approx \sum_{k=0}^{5} E^{n\delta t}_{z_{0,5,k}} \delta z_k \tag{45.6}$$

The current is obtained in a similar way by integrating numerically the magnetic field along the conductor,

[1] Hollow waveguides need a special treatment

where the path of integration is specified by the box at $i = 1/2$ and $k = 5$:

$$\begin{aligned} i(t) = \oint_C \vec{H} \cdot \mathrm{d}\vec{\ell} \;\approx\; & \sum_{j=3}^{7} H_{\mathrm{y}_{0,j,5}}^{(n+1/2)\delta t} \frac{\delta y_j + \delta y_{j-1}}{2} \\ - \; & \sum_{j=3}^{7} H_{\mathrm{y}_{0,j,4}}^{(n+1/2)\delta t} \frac{\delta y_j + \delta y_{j-1}}{2} \\ - \; & H_{\mathrm{y}_{0,7,5}}^{(n+1/2)\delta t} \frac{\delta z_5 + \delta z_4}{2} \\ + \; & H_{\mathrm{y}_{0,3,5}}^{(n+1/2)\delta t} \frac{\delta z_5 + \delta z_4}{2} \end{aligned} \tag{45.7}$$

45.1.2. Lumped ports

Another type of port is featured by EMPIRE XPU™ which is called lumped port. This port can be placed between two conductors and excites the electric field between them. Voltages and currents are recorded at the load location.

For this, a port can be defined in a similar way as done as depicted in Figure 45.3.

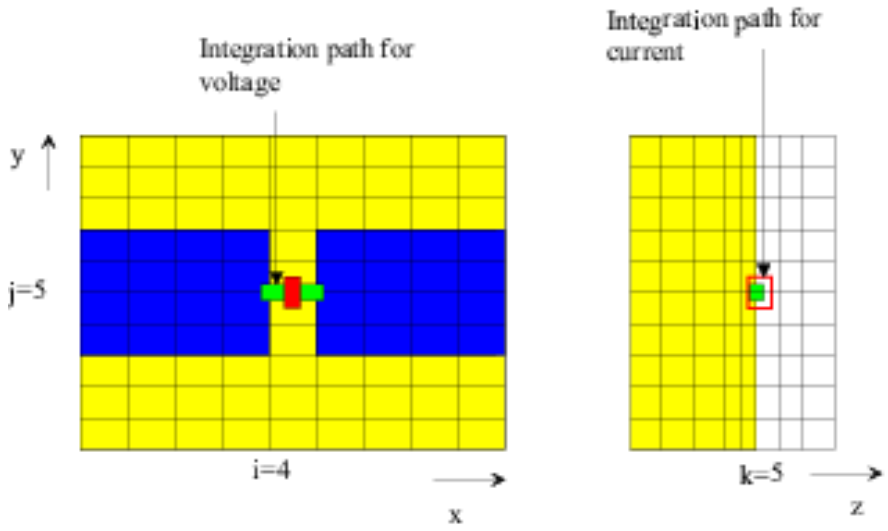


Figure 45.3.: Illustration of an inner port

By default, the characteristic reference impedances are set to 50Ω.

45.1.3. Waveguide ports

To simulate waveguide ports, the modes have to be pre-calculated by running a two-dimensional eigenvalue solver. The shape may be arbitrary but must be homogeneously filled with material as depicted in Figure 45.4.

Hint

Magnetic may not be used for waveguide port calculation.

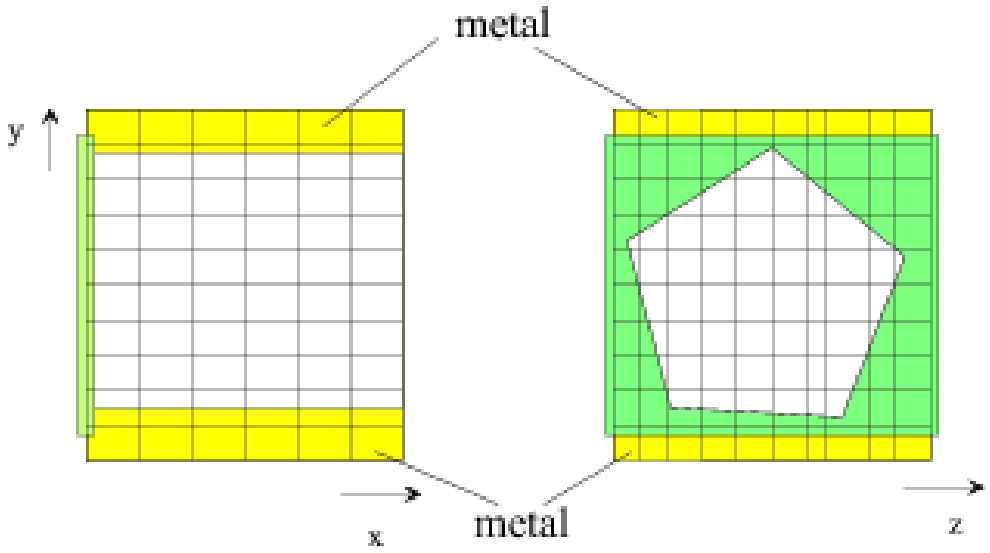


Figure 45.4.: Illustration of a waveguide port

An eigenvalue solver calculates a set of orthogonal eigenvectors which are normalized. For example, modes in the plane $x_{\min}$ can be expressed as

$$\hat{e}_{n_{j,k}} = \frac{\vec{e}_{n_{j,k}}}{\sqrt{\sum_j \sum_k \vec{e}_{n_{j,k}} \cdot \vec{e}_{n_{j,k}} \delta y_j \delta z_k}} \tag{45.8}$$

$$\hat{h}_{n_{j,k}} = \frac{\vec{h}_{n_{j,k}}}{\sqrt{\sum_j \sum_k \vec{h}_{n_{j,k}} \cdot \vec{h}_{n_{j,k}} \delta y_j \delta z_k}} \tag{45.9}$$

Although voltages and currents can not be uniquely defined for waveguides, it is convenient to record the following expressions:

$$u_n(t) = \sum_j \sum_k \vec{E}^n_{\mathrm{t}_{0,j,k}} \cdot \hat{e}_{n_{j,k}} \delta y_j \delta z_k \tag{45.10}$$

$$i_n(t) = \sum_j \sum_k \vec{H}^{n+1/2}_{\mathrm{t}_{0,j,k}} \cdot \hat{h}_{n_{j,k}} \delta y_j \delta z_k \tag{45.11}$$

Here, the subscript t denotes the transverse field values. According to the specific mode n, equivalent voltages and currents are recorded during the simulation run. After a discrete Fourier transformation, (45.3,45.4) can be applied to get the desired wave quantities.

45.2. Time signals

During the simulation run, currents and voltages are recorded. It is possible to obtain the values for every time step. But for some reason the signals can be highly oversampled[2]. Thus, before the simulation run, a proper sampling factor n_{sample} will be determined, and only the necessary number of values are stored.

$$u(t) = u(n_{\mathrm{data}} n_{\mathrm{sample}} \delta t) \tag{45.12}$$

where n_{data} is the actual data point in the data file ut.

[2] if the time step is extremely small compared to the period of the upper frequency limit

45.2.1. Absorbing Port signal processing

As an example, for a 3 port simulation, excited at port 1 with a matched source, the following time signals will be obtained:

$$ut1.inc \quad ut1 \quad ut2 \quad ut3 \tag{45.13}$$
$$it1.inc \quad it1 \quad it2 \quad it3 \tag{45.14}$$

The reflected waves are simply obtained by

$$ut1.ref = ut1 - ut1.inc \tag{45.15}$$
$$it1.ref = it1 - it1.inc \tag{45.16}$$

45.2.2. Concentrated Port signal processing

As an example, for a 3 port simulation, excited at port 1 with a concentrate source, the following time signals will be obtained:

$$ut1 \quad ut2 \quad ut3 \tag{45.17}$$
$$it1 \quad it2 \quad it3 \tag{45.18}$$

The wave separation is obtained by a Heaviside transformation with the internal port impedance R_{lpi} which has to coincide with the value adjusted in the port settings.

$$ut1.inc = \frac{1}{2}ut1 + \frac{R_{lpi}}{2}it1 \tag{45.19}$$
$$it1.inc = \frac{1}{2}it1 + \frac{1}{2R_{lpi}}ut1 \tag{45.20}$$

$$ut1.ref = ut1 - ut1.inc \tag{45.21}$$
$$it1.ref = it1 - it1.inc \tag{45.22}$$

45.3. Discrete Fourier Transformation of signals

After the simulation run, the DFT of the time signals can be carried out to get the desired quantities in frequency domain. In example for the voltages the following relation is applied.

$$u(f) = \sum_{n=0}^{N_{\text{data}}} u(t)e^{-2\pi f n n_{\text{sample}}\delta t}\delta t \tag{45.23}$$

As an example, for a 3 port simulation, excited at port 1, the following frequency dependent parameters will be generated automatically by a DFT with the data file names:

$$uf1.inc \quad uf1.ref \quad uf1 \quad uf2 \quad uf3 \tag{45.24}$$
$$if1.inc \quad if1.ref \quad if1 \quad if2 \quad if3 \tag{45.25}$$

45.4. Impedances

According to (45.5), the impedances will be calculated. As an example, for a 3 port simulation, excited at port 1, the impedances are obtained by:

$$Z1 = \frac{uf1.inc}{if1.inc} \tag{45.26}$$

$$Z1.in = \frac{uf1}{if1} \tag{45.27}$$

$$Z2 = \frac{uf2}{if2} \tag{45.28}$$

$$Z3 = \frac{uf3}{if3} \tag{45.29}$$

where the impedance seen from the exciting port is denoted $Z1.in$

45.5. Wave quantities

According to (45.4), the wave quantities will be calculated. As an example, for a 3 port simulation, excited at port 1, the following values are generated:

$$b_1 = \frac{uf1.ref}{\sqrt{Z1}} \tag{45.30}$$

$$b_2 = \frac{uf2}{\sqrt{Z2}} \tag{45.31}$$

$$b_3 = \frac{uf3}{\sqrt{Z3}} \tag{45.32}$$

The incident wave quantities are set to:

$$a_1 = 1 \tag{45.33}$$

$$a_2 = 0 \tag{45.34}$$

$$a_3 = 0 \tag{45.35}$$

45.6. Scattering parameters

According to (45.1), the scattering parameters will be calculated. As an example, for a 3 port simulation, excited at port 1, one column of the scattering matrix is generated:

$$s_{11} = \frac{b_1}{a_1} \tag{45.36}$$

$$s_{21} = \frac{b_2}{a_1} \tag{45.37}$$

$$s_{31} = \frac{b_3}{a_1} \tag{45.38}$$

For a complete matrix, the simulation has to be carried out as many times as ports exist. This will be done automatically.

46. Specials

This chapter deals with some theoretical background of special features of the EMPIRE XPU™ program.

46.1. Y-Parameter Processing

S-Parameter processing as defined before assumes a single characteristic impedance which is valid for the complete frequency range. Actually, impedances are frequency dependent and the termination is not ideal in any case. Thus the port properties may influence the S-Parameter results.

In order to minimize the effects of the ports an alternative processing can be applied. A prerequisite is that all ports $i = 1, ..., N$ are excited and each result folder $sub - i$ contains the frequency domain voltages $\underline{\underline{U}} = uf_{ij}$ and currents $\underline{\underline{I}} = if_{ij}$ with $j = 1, ..., N$. Then the admittance matrix can be obtained:

$$\underline{\underline{Y}} = \underline{\underline{I}}\underline{\underline{U}}^{-1} \tag{46.1}$$

With the diagonal characteristic admittance matrix $Y_{ii}^L = 1/Z_{ii}^L$ and the normalization

$$N_{ij} = \sqrt{\frac{Y_{ii}^L}{Y_{jj}^L}} \tag{46.2}$$

the S-Matrix can be obtained with

$$\underline{\underline{S}} = \underline{\underline{N}}(\underline{\underline{Y^L}} + \underline{\underline{Y}})^{-1}(\underline{\underline{Y^L}} - \underline{\underline{Y}}) \tag{46.3}$$

46.2. Hollow Waveguides

For a given frequency f, only a finite number of modes can propagate inside a homogeneously filled conducting cylinder with arbitrary cross section. This number increases with the frequency, implying that every hollow waveguide mode has a cut-off frequency f_c. Below f_c the mode is evanescent and above f_c the mode can propagate along the cylinder having a well known propagation characteristic (44.3), where, c_0 denotes the velocity in free space. Two types of waveguide modes can be distinguished, transversal electric (TE) modes and transversal magnetic (TM) modes, having only a magnetic and an electric component in propagation direction, respectively. They fulfill the two-dimensional Helmholtz equation (46.4).

$$\Delta_t H_z + \left(\frac{2\pi f_c}{c_0}\right)^2 H_z = 0 \tag{46.4}$$

$$\Delta_t E_z + \left(\frac{2\pi f_c}{c_0}\right)^2 E_z = 0 \tag{46.5}$$

By discretizing equations (46.4) according to the Finite Difference scheme and employing the appropriate boundary conditions, the eigenvalue problem for f_c can be solved numerically using a two-dimensional Finite Difference Frequency Domain (FDFD) solver, and eigenvectors can be found for H_z and E_z for TE-modes

and TM-modes, respectively. The transversal structure functions for these modes, which are independent of frequency for homogeneously filled waveguides, can be derived from the longitudinal components H_z and E_z, leading to the desired eigenfunctions (46.6).

$$\vec{e}_{n_{i,j}} = \nabla_t H_z \tag{46.6}$$
$$\vec{h}_{n_{i,j}} = \nabla_t E_z \tag{46.7}$$

46.3. Near to Far Field Transformation

The near-field around a radiator or a scatterer can be sampled and saved for a set of frequency points during the simulation run. After simulation the field can be transformed to the far-field resulting in radiation pattern of mono-static (RCS) or bi-static cross section.

By introducing electric and magnetic current densities (46.8) derived from the field values on the near-field to far-field box the integrals in (46.10) are solved numerically in the required observation direction.

$$\vec{S} = \hat{n} \times \vec{H} \tag{46.8}$$
$$\vec{M} = -\hat{n} \times \vec{E} \tag{46.9}$$

$$N_\theta = \iint (S_x \cos\theta \cos\phi + S_y \cos\theta \sin\phi - S_z \sin\theta) e^{jkr' cos\Psi} ds' \tag{46.10}$$
$$N_\varphi = \iint (-S_x \sin\phi + S_y \cos\phi) e^{jkr' cos\Psi} ds' \tag{46.11}$$
$$L_\theta = \iint (M_x \cos\theta \cos\phi + M_y \cos\theta \sin\phi - M_z \sin\theta) e^{jkr' cos\Psi} ds' \tag{46.12}$$
$$L_\varphi = \iint (-M_x \sin\phi + M_y \cos\phi) e^{jkr' cos\Psi} ds' \tag{46.13}$$

In these equations Ψ is the angle between observation point and source point.

The far-field is then obtained using (46.14).

$$\bar{E}_\theta(\theta, \varphi) = \frac{-k}{\sqrt{8\pi P Z_0}} (L_\varphi + Z_0 N_\theta) \tag{46.14}$$
$$\bar{E}_\varphi(\theta, \varphi) = \frac{k}{\sqrt{8\pi P Z_0}} (L_\theta - Z_0 N_\varphi) \tag{46.15}$$

The value for P is chosen depending on the kind of normalization:

- Maximum normalization. All values are divided by the maximum.
- Directivity normalization. P is the radiated power from the source, evaluated by Poynting vector integration over the near-field.

46.4. SAR and ACD

The distribution of the specific absorption rate SAR and current density ACD $\vec{S}$ are determined by the distribution of the electric field $\vec{E}$, the mass density ρ and the electric conductivity σ.

$$\vec{J} = \sigma\vec{E} \tag{46.16}$$

$$SAR = \frac{\sigma|\vec{E}|^2}{\rho} \tag{46.17}$$

The SAR and current density values are important field quantities, especially in the field of safety issues and numerical dose assessment. Numerous national and international standards and guidelines in the field of human exposure to electromagnetic fields claim for the compliance with so-called *basic restrictions*. In the high-frequency range this basic restrictions are given in terms of specific absorption rates and in the low-frequency range this basic restrictions are given in terms of current densities. As far as dose assessment is concerned the evaluation of compliance with standards and limits is assisted by the software. For this the local SAR and ACD values are averaged and the maximums are determined [5], [6]. The SAR is averaged over a mass of 1 g or 10 g. The ACD is averaged over an area of 1 cm^2 or 100 cm^2.

Hint: SAR averaging

EMPIRE XPU™ applies an averaging method according to IEEE/IEC 62704, "Recommended Practice for Determining the Peak Spatial-Average Specific Absorption Rate (SAR) in the Human Body from Wireless Communications Devices, 30 MHz - 6 GHz"

For compatibility reasons older standards, like IEEE/P1528 and IEEE/C95.3 can still be selected.

46.5. Low-Frequency Algorithm

In the field of human exposure to low-frequency electromagnetic fields (e.g. power line frequency) the determination of induced electric fields inside the human body is of great importance. The application of the FDTD-method in the calculation of low-frequency fields inside the human body results in long runtime. Therefore an algorithm presented in the literature is applied in order to calculate the induced fields more efficiently.

The implemented algorithm has been proposed by *O.P. Gandhi* and is capable of calculating induced currents in lossy dielectric material on condition, that the displacement current density is negligible ($\omega\varepsilon \ll \sigma$). Considering this prerequisite the simulation is performed with a permittivity of free space ($\varepsilon_r = 1$) in order to reduce the simulation time. The propagation speed of an electromagnetic wave inside the human body decreases with increasing ε_r. Therefore, a high permittivity would result in a low propagation speed and long runtime.

The algorithm of O.P. Gandhi takes advantage of the quasi-static nature of low-frequency fields. As a result the actual simulation is performed at a higher frequency (with the quasi-static approximation being still valid) and the induced fields are *scaled* to the frequency of interest after the higher frequency FDTD-simulation has been finished. For typical tissue parameters and dimensions of the human body the upper limit is about 5 MHz to 10 MHz. The calculation of the induced low-frequency fields down to 10 Hz is accomplished by scaling the high-frequency results. The method makes use of the boundary condition of the electric field in the air $\vec{E}_{Air}$ and the electric field inside the human body $\vec{E}_{Body}$

$$j\omega\varepsilon_0\vec{n}\vec{E}_{Air} = (\sigma + j\omega\varepsilon)\vec{n}\vec{E}_{Body} \tag{46.18}$$

and the approximation of a normal orientation of the electric field on the surface of a conducting body. The induced electric field for the angular frequency of interest ω is determined by the scaled results at an angular frequency of ω'.

$$\vec{E}_{Body}(\omega) = \frac{\omega}{\omega'} \frac{\sigma' + j\omega'\varepsilon'}{\sigma + j\omega\varepsilon} \vec{E}_{Body}(\omega') \tag{46.19}$$

46.6. Dielectric Properties of Biological Tissue

In order to consider the frequency-dependent dielectric properties of biological tissue a parametric model from literature[1] has been implemented. The model provides the relative permittivity ε_r and the electric conductivity σ of 42 different human tissue types in the frequency range from 10 Hz up to 100 GHz.

The complex permittivity is expressed by the following equation (46.20). The coefficients are summarized in the cited literature.

$$\underline{\varepsilon}(\omega) = \varepsilon_\infty + \sum_{n=1}^{4} \frac{\Delta\varepsilon_n}{1 + (j\omega\tau_n)^{1-\alpha_n}} + \frac{\sigma}{j\omega\varepsilon_0} \tag{46.20}$$

The parametric model has been integrated in the user interface in order to consider the relevant dielectric parameters in the simulation model in an easy way.

[1] S. Gabriel, R. W. Lau, C. Gabriel. The dielectric properties of biological tissue: III. Parametric models for the dielectric spectrum of tissues. Phys. Med. Biol., Vol. 41, p. 2271-2293, 1996.

Part VII.

Literature

Bibliography

[1] W.Simon, A.Lauer, D.Manteuffel, A.Wien, and I.Wolff, "Efficient FDTD parallel processing on modern PC CPUs," in *EuroEM Conference*, (Magdeburg), pp. 0–0, 2004.

[2] B. Derat, M. Celik, W. Simon, and A. Lauer, "Evaluation of integrated antenna performance through combined use of measurement and full-wave simulation," in *2020 Antenna Measurement Techniques Association Symposium (AMTA)*, (Online), pp. 1–5, 2020.

[3] K. S. Yee, "Numerical solution of initial boundary value problems involving Maxwell's equations in isotropic media," *IEEE Trans. Antennas Propagat.*, vol. AP-14, pp. 302–307, 1966.

[4] R. Mittra and U. Pekel, "A new look at the Perfectly Matched Layer (PML) concept for the reflectionless absorption of electromagnetic waves," *IEEE Microwave and Guided Wave Lett.*, vol. 5, pp. 84–86, 1995.

[5] D.Manteuffel, D.Heberling, and W.Simon, "Numerical dosimetric assessment using the FDTD method," in *ITG Conference INICA*, (Berlin), pp. 0–0, 2003.

[6] W.Simon, D.Manteuffel, and A.Lauer, "New techniques for FDTD calculation of the SAR induced in the human head by wireless communication devices," in *Loughborough Antennas and Propagation Conference*, (Loughborough), pp. 0–0, 2005.

[7] A. Lauer and I. Wolff, "Stable and efficient ABCs for graded mesh FDTD simulations," *IEEE MTT-S Int. Microwave Symp. Digest*, vol. 2, pp. 461–464, 1998.

[8] A. Lauer and I. Wolff, "A conducting sheet model for efficient wide band FDTD analysis of planar waveguides and circuits," *IEEE MTT-S Int. Microwave Symp. Digest*, vol. 4, pp. 1589–1592, 1999.

[9] A. Lauer and I. Wolff, "Improved autoregressive (AR) signal modeling for FDTD resonance estimation," *IEEE MTT-S Int. Microwave Symp. Digest*, vol. 1, pp. 255–258, 2000.

[10] A. Wien, A. Lauer, and I. Wolff, "Equivalent circuit generation in FDTD package modeling," in *IMAPS Symposium Digest*, (Boston), pp. 255–258, 2000.

[11] A. Lauer, A. Wien, P. Waldow, and I. Wolff, "Enhanced PML-like ABCs for layered media transmission line termination," in *IEEE MTT-S Int. Microwave Symp. Digest*, (Seattle), pp. 739–742, 2002.

[12] A. Wien, A. Lauer, and I. Wolff, "Multi-PC FDTD: Solving large scale EM problems," in *IEEE MTT-S Int. Microwave Symp. Digest*, (Anaheim), pp. 972–975, 2010.

Part VIII.

Index

List of Actions


2D Scale to Origin 444

Add Fixed 432

Add Fixed Lines 433

Add Port Number(s) 430

Add Property 442

Add To Discretization 433

Add Unfixed Lines 433

Align Object to Arrow 445

Align Object to Point 445

Area Measurement 425

Arrow 1/3rd Rule 434

Arrow Concat (+) 422

Arrow Multiple (&) 422

Arrow Scale (*) 423

Arrow Subdivide (/) 423

Automesh Setup 408

Bisect Polygon 439

Change Origin 416

Change Sign 414

Change To Poly 415

Change to Line poly (Script) 431

Check Distance 425

Check Integrity 426

Check Polygon Point Spacing 426

Check Symmetry 426

Circle Recognition 440

Convert Multibox to Boxes 428

Convert to Bounding Box 428

Convert to Poly 428

Convert to Solid 428

Copy (*) 418

Copy Marked 436

Copy & Mirror (bar) 419

Copy & Rotate 420

Copy and Edit Box 424

Copy to Current Group 430

Create Arrow (-) 423

Create Bond Wire 432

Create Box 402

Create Circuit Element 406

Create Connectivity Groups 426

Create Connectivity Groups (discretized) . 426

Create Cylinder (Circular) 403

Create Dimensioning 424

Create Field Monitor 406

Create Library Object 404

Create Line Poly 432

Create Mesh 413

Create Polygon 403

Create Source 405

Cycle Elements 440

Delete (Del) 410

Delete Fixed Range 435

Delete Last Property 442

Delete Marks 438

Delete Mesh 407

Delete Mesh 435

Delete Mesh Filling 407

Delete Points 416

Delete Properties 442

Delete Range 435

Deselect Enclosed (D) 443

Deselect Overlapping (d) 443

Edit Box 424

Edit Parameters 410

Edit Point/Arrow 412

Edit Properties 442

Edit Substructure 412

Edit Text 415

Editor Options 409

Escape (Esc) 402

Explode Library Element 430

Extract Points 414

Extrude Flat Solid 444

Fix Range 436

Front View 422

Fullscreen 421

Generic Discretization 437

Group Boxes 428

Import 412

Insert Point(s) 416

Insert Symmetric 413

 Intersect (Ctrl y) 401
 Iso X View 422
 Iso Y View 422
 Iso Z View 422
 Last Selection(l) 417
 Length Measurement 427
 Linpoly to Solid 432
 Map Object on Mesh 427
 Mark All 438
 Mark Range 438
 Merge 401
 Mesh Info 427
 Meshed 3D Structure 414
 Mirror (&) 419
 Move (+) 418
 Move Parameter 439
 Move to Current Group 430
 Multiple Copy 418
 Multiple Subtract 401
 Negate (m) 423
 Open Current Group 413
 Open Object (o) 410
 Optimize Multibox 429
 Optimize Solid 1 429
 Optimize Solid 2 429
 Oversize 445
 Oversize with Roundings 445
 Poly to Rounded Strip 440
 Poly to Strip 440
 Poly to Wire 441
 Propose Discretization 434
 Propose Discretization (local) 434
 Push Regular Expression 415
 Rasterize Polygon 441
 Reciprocal 415
 Redo 411
 Redraw 409
 Restore View 409
 Return Center Point 423
 Revolve Polygon around Axis 416
 Rotate 419
 Rotate Structure and Mesh 445
 Save (Ctrl-s) 412
 Save View 409
 Save & Compile ..... ??
 Scale to Object Center 444
 Scale to the Origin 444
 Select Enclosed (e) 417
 Select Groups' Objects 430
 Select Outside 443
 Select Overlapping (p) 417
 Set Height of Current Group 430
 Set Height(=) 421
 Set Parameter by Equation 411
 Set Parameters by File 439
 Set Port Number(s) 410
 Set Property 442
 Set Radius 441
 Set Tangential 432
 Shift Marked 436
 Show Exploded Library Element 431
 Show Mapped Object 427
 Side View 421
 Simplify Polygon 429
 Simulation Setup 408
 Smooth Discretization 437
 Smooth Discretization Marked 437
 Split Solid 429
 Start Simulation 408
 Store Variant 408
 Stretch (/) 420
 Stretch Parameter 439
 Subdivide 414
 Subtract first-second 401
 Subtract Port Number(s) 431
 Toggle Mode 415
 Top View 422
 Undo 411
 Unfix Range 436
 User Vertex Transform 443
 Via Fence 425
 Zoom Extents (z) 408
 Zoom Window (z) 409

List of Options

2D Draft Display 458
3D Display Linewidth 455
3D Measure Cone size 463
3D Measure Distance 463
3D Measure Font size 463
3D Measure Line Width 463
Accept Zero Area Objects 469
Arc Recognition min. # Segments 464
Arc Resolution (DEG) 468
Arrow Distance 462
Arrow Heads 462
Arrow Rectangle Color 447
Arrow Text 447
Arrow color 447
Automatic Circles 456
Background Color 450
Background Color 454
Background Gradient 454
Circle Recognition Points 469
Circle Recognition Tolerance 469
Circle Recognition min. # Points 457
Circle Recognition min. # Points 464
Circle Resolution (Degrees) 456
Circle Tolerance 464
Circle diameter Scale 467
Circle min # points 466
Circle min # points 467
Circle tolerance 466
Circle tolerance 467
Coordinate Shift 468
Create Backup Files 472
Cursor Color 448
Cursor Size 448
DRC Failure Behavior 461
DXF Connection Tolerance 469
DXF General Tolerance 469
DXF Scale 464
Design Check Tolerance 461
Dimensioning Number of Digits 462
Dimensioning Scale Factor 462
Dimensioning Tic Size 462
Display Grid 451
Display Mesh 451
Display Mode 454
Distance to Poly 459
Double Via Holes 467
EXCELLON decimal digits 471
Font Size 463
GDS VIA Distance (GDS units) 470
GDS VIA Size (GDS units) 470
GDS min. Obj size (GDS units) 470
GDSII Scale 464
GERBER Aperture Size (mm) 465
GERBER Float Format 465
GERBER Merge Tolerance 470
GERBER Scale 464
GYM Scale Factor 471
GYM Shift (x y z) 471
Galvo speed 467
Group Drawing Order 450
HPGL pen width 471
HPGL scale 466
Horizontal Component Snap 447
Horizontal Mesh Snap 449
Horizontal Snap 449
Icon Size 472
Ignore layer in DXF block 469
Import Scale Factor 468
Lamp Current (A) 468
Laser off speed 467
Laser on speed 467
Light 0 Ambient 452
Light 0 Diffuse 453
Light 0 Position 452
Light 0 Specular 453
Light 1 Ambient 453
Light 1 Diffuse 453
Light 1 Position 453
Light 1 Specular 453
Light Global Ambient 453
Line Poly Default Width 456
Linewidth 452
Logging 461
Max Circle diameter 467
Max Number of Solid Handles 458
Max instructions per file 467
Max. Number of Handles 452
Max. Number of curves with full table display 451
Maximum Iterations 460
Merge GERBER Layers 470
Min. Normal Angle for Wireframe 451
Min. Normal Angle in 2D display 457
Minimum Auto Thickness 461
Multiple Copy Expression 462
Number of Digits 448
Number of Points in Bond Wire 455
Output Format 466
POSTSCRIPT Page Width 468
PS Layout Scale 468
PS Layout shift 468
Pad Group(s) 460
Pad Width 459
Point Text 450
Point color 450
Point size 450

Pointsize 452
Poly Minimum Angle 455
Poly Rasterize 455
Poly Set Radius Option 456
Poly Tolerance 467
Poly tolerance 457
Pulse Frequency (kHz) 468
Pulse Length (s) 468
Relative Accuracy 458
Repetitions 466
Resolution 454
Resolution Factor 463
Resolution Factor 465
Rotation Angle 461
Rotpoly Fast Discretization 457
Rotpoly Subdivision (Integer) 456
STL Scale 465
STL Shift (x y z) 470
STL Simplification 471
Save all Triangles in ASCII 458
Scale Factor 466
Scaling Factor 466
Separate STL Files 465
Separate STL Objects 470
Show Bounding Box 457
Show Coordinate System 465
Show Mesh Coordinate 448
Slope Angle phi0= 455
Slope Angle phi1= 455
Snap Mode 447
Snap on distant Components 448
Snapped Cursor Color 449
Software 3D antialiasing 454
Speed in ms 452
Start Coordinates 466
Start Coordinates 466
Subroutine Optimization 467
Transparency Setup 454
Undersize 467
Unit 466
Use Attractors 460
Use slow DRC check 460
Vertical Component Snap 448
Vertical Mesh Snap 449
Vertical Snap 449
Via A Group(s) 459
Via B Group(s) 460
Via Diameter 459
Via Distance 459
Via # Points 460
Warn for Improper Objects 458
Wheel Zoom Factor 450

List of Autodisc Options

Absolute min in units 475
Arc Resolution in Degrees 475
Edge accuracy (Percent) 477
Flat Metal Edge correction 473
Flat metal Recognition Limit 477
Max Resolution in units 475
Max. scan points per Object 477
Mesh Resolution (Numeric) 473
Min Cells per Wavelength 474
More Lines between Edges 473
Object Roughness (cells) 474
Object Scan Angle (Degrees) 473
Object cells 1 476
Object cells 2 476
Object cells 3 476
Object cells 4 476
Planar Direction 475
Refinement ratio 476
Relative Drawing Accuracy 474
Relative Limit for Warning 474
Resolve Curved Geometry 473
Target min in units 475
User Defined Simulation Box 479
Wedge limit in Degrees 477
max 479
max 479
max. Aspect ratio 477
min 479
min 479
x mesh in units 477
x mesh options in units 478
y mesh in units 478
y mesh options in units 478
z mesh in units 478
z mesh options in units 478
zmax 480
zmin 479

List of Simulation Options

AC only simulation 481
Accuracy 489
Algorithm 488
Ambient Temperature 489
Blood Density kg/m**3 490
Blood Heat Capacity J/kg/K 490
Blood Temperature in K 490
Conductors 487
Convection 490

Dielectrics 487
Drawing Unit 486
End Frequency 485
Energy Decay 481
Field Convergence Check 481
Frequencies 483
Maximum Temperature 489
Mesh Resolution 486
Mode 488
Number for Frequency Points 485
PGA 481
Permittivity 483
Radiation 491
Resolve Metal sheets 488
Sides on Ambient Temperature 490
Simulation Mode 489
Solvers 486
Start Frequency 485
Structure Type 486
Target Frequency 485
Temperature Storage 489
xmax 482
xmax 484
xmin 482
xmin 483
ymax 482
ymax 484
ymin 482
ymin 484
zmax 483
zmax 484
zmin 482
zmin 484