

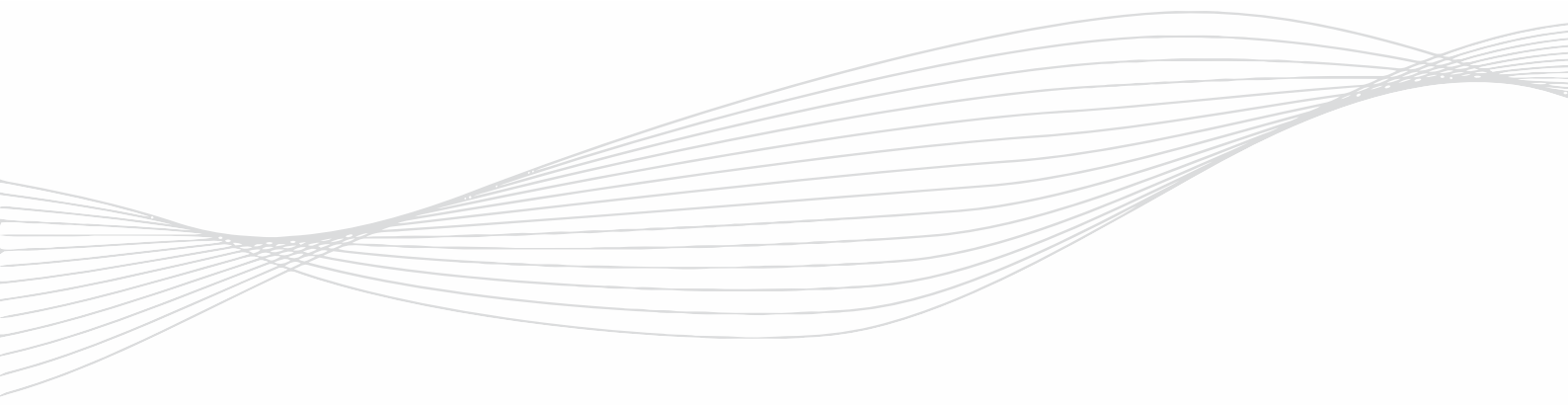
# Empire XPU Manual



Version 9.0.0

(c) IMST GmbH 1998-2024

March 13, 2024



# Contents

<b>I. Software Administration</b>	<b>1</b>
<b>1. Installation on Windows PC</b>	<b>2</b>
1.1. Requirements for PC Installation	2
1.2. Installation Procedure	2
1.3. Multiple version installations	6
1.4. Silent installation	6
1.5. Uninstall	6
<b>2. License Setup</b>	<b>7</b>
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
<b>3. Starting Empire on Windows</b>	<b>12</b>
3.1. Simple Test Procedure	12
3.2. Graphic Renderer Information	15
<b>4. Floating License on Windows</b>	<b>17</b>
4.1. Floating License File	17
4.2. Installation on the Server	18
4.2.1. Delayed license checkout	20
4.3. Installation on each Client	20
4.4. License Diagnostic	21
<b>5. Remote Control</b>	<b>23</b>
5.1. Remote Desktop	23
5.1.1. Windows Remote desktop	23
5.1.2. PCoIP	24
5.2. Remote computing	24
5.2.1. Starting the remote service in EMPIRE XPU™	25
5.2.2. Starting the remote service as a Windows command	26
5.2.3. Starting the remote service as a Windows service	26
5.2.4. Batch processing using IP addresses	28
<b>6. Multi-PC Solver</b>	<b>29</b>
<b>7. Virtual Model Inspection Module</b>	<b>31</b>
<b>8. Linux systems</b>	<b>32</b>
8.1. Requirements	32
8.2. Installation	32
8.3. Start Commands	32
8.3.1. Background processing	33

8.4. License Request . . . . .	33
8.5. Install the License . . . . .	34
8.6. Floating license . . . . .	34
8.6.1. Delayed license checkout (Linux) . . . . .	34
8.7. Trouble Shooting . . . . .	35
<b>9. Data Structure</b>	<b>36</b>
9.1. Input Files . . . . .	36
9.1.1. File Recovery . . . . .	36
9.1.2. Compatibility . . . . .	36
9.2. Simulation Files . . . . .	36
9.3. Postprocessing Files . . . . .	37
<b>10. High Performance Computing</b>	<b>38</b>
10.1. Single machine consideration . . . . .	38
10.2. Multiple machine considerations . . . . .	39
10.2.1. Multiple jobs on different machines (job distribution) . . . . .	39
10.2.2. Single jobs on different machines (job sharing) . . . . .	39
<b>11. Update &amp; Support</b>	<b>41</b>
11.1. Version Control . . . . .	41
11.2. Web Sites . . . . .	41
11.3. Contact Us . . . . .	41
11.4. Contact Local Support Teams . . . . .	42
 <b>II. Empire User Interface</b>	 <b>43</b>
<b>12. Introduction</b>	<b>44</b>
12.1. Top tool bars . . . . .	45
12.1.1. Menu items . . . . .	45
12.1.2. Application tabs . . . . .	49
12.1.3. Top left icons . . . . .	49
12.2. 3D Design Mode . . . . .	52
12.2.1. Grid . . . . .	52
12.2.2. Cursor . . . . .	54
12.2.3. Coordinate Systems . . . . .	54
12.2.4. Zoom . . . . .	56
12.2.5. Pan . . . . .	56
12.2.6. Rotate . . . . .	56
12.2.7. Bottom line . . . . .	57
12.3. 2D Design Mode . . . . .	57
12.3.1. Background . . . . .	57
12.3.2. Zoom . . . . .	58
12.3.3. Pan . . . . .	58
12.3.4. Origin, WCS . . . . .	58
12.3.5. Simulation domain . . . . .	58
12.3.6. Bottom line . . . . .	59
12.4. 3D Results Mode . . . . .	59
12.4.1. Background . . . . .	59
12.4.2. Zoom . . . . .	60
12.4.3. Pan . . . . .	60
12.4.4. Rotate . . . . .	60
12.4.5. Display Buttons . . . . .	60

12.4.6. Context Menu . . . . .	61
12.5. Operation buttons . . . . .	62
12.6. Side menu / Navigation tree . . . . .	64
<b>13. Operators, Selections and Shortcuts</b>	<b>66</b>
13.1. Operators . . . . .	66
13.2. Object types . . . . .	66
13.3. Select objects . . . . .	67
13.4. Handles . . . . .	67
13.5. Deselect objects . . . . .	67
13.6. Shortcuts . . . . .	68
<b>14. Groups</b>	<b>69</b>
14.1. Control for all groups . . . . .	70
14.2. Control for single groups . . . . .	71
<b>15. Object creation</b>	<b>73</b>
15.1. Basic Objects . . . . .	73
15.1.1. Box . . . . .	74
15.1.2. Circular Cylinder . . . . .	74
15.1.3. Polygon . . . . .	75
15.1.4. Strip . . . . .	77
15.2. Object Library . . . . .	78
15.2.1. 3D Solid . . . . .	79
15.2.2. 3D Surface . . . . .	81
15.2.3. 3D Wire . . . . .	83
15.2.4. 2D Extruded . . . . .	83
15.2.5. Coax . . . . .	85
15.2.6. MSL Planar . . . . .	88
15.2.7. CPW Planar . . . . .	88
15.2.8. SMDs . . . . .	90
15.2.9. Substructure . . . . .	91
15.2.10. Layout . . . . .	95
15.2.11. Conformal . . . . .	95
15.2.12. User . . . . .	96
<b>16. Object modification</b>	<b>97</b>
16.1. Move Operations . . . . .	97
16.1.1. Move Object . . . . .	97
16.1.2. Move Object to Group . . . . .	97
16.1.3. Parametric Move . . . . .	98
16.1.4. Mirror . . . . .	98
16.1.5. Rotate . . . . .	98
16.1.6. Align . . . . .	99
16.1.7. Transform via LCS . . . . .	99
16.2. Copy Operations . . . . .	99
16.2.1. Copy Object . . . . .	99
16.2.2. Copy to Current Group . . . . .	100
16.2.3. Array Copy . . . . .	100
16.2.4. Copy & Mirror . . . . .	101
16.2.5. Copy & Rotate . . . . .	101
16.3. Stretch Operations . . . . .	101
16.3.1. Stretch Object . . . . .	101
16.3.2. Parametric Stretch . . . . .	102

16.3.3. Scale . . . . .	102
16.3.4. Oversize . . . . .	103
16.3.5. Rasterize Polygons . . . . .	103
16.4. Change operations . . . . .	103
16.4.1. Set radius . . . . .	103
16.4.2. Convert Boxes . . . . .	103
16.4.3. Convert Polygons . . . . .	104
16.4.4. Convert Solid . . . . .	104
16.5. Boolean operations . . . . .	104
<b>17. Import and Export</b>	<b>106</b>
17.1. Import . . . . .	106
17.1.1. Advanced . . . . .	106
17.1.2. Empire . . . . .	107
17.1.3. EDA . . . . .	107
17.1.4. 2D Layout Data . . . . .	108
17.1.5. 3D Solid Data . . . . .	109
17.1.6. CST-Projects . . . . .	112
17.1.7. HFSS-Projects . . . . .	114
17.1.8. Data - Import Touchstone . . . . .	115
17.2. Export . . . . .	115
17.2.1. Advanced . . . . .	115
17.2.2. 2D Layout . . . . .	116
17.2.3. 3D Solids . . . . .	116
17.2.4. Empire Part . . . . .	117
17.2.5. Read Protected File . . . . .	117
17.2.6. Export Image . . . . .	117
<b>18. Object properties</b>	<b>118</b>
18.1. Priorities . . . . .	118
18.2. Material Property . . . . .	119
18.2.1. Dielectric . . . . .	119
18.2.2. Conductor . . . . .	120
18.2.3. Absorber . . . . .	122
18.2.4. Advanced - Material Object . . . . .	124
18.2.5. Advanced - Debye Material . . . . .	125
18.2.6. Advanced - Drude Plasma . . . . .	127
18.2.7. Advanced - Meta Material . . . . .	128
18.2.8. Advanced - Gabriel Material . . . . .	129
18.2.9. Advanced - Conformal Dielectrics . . . . .	132
18.2.10. Advanced - Metal Sheet Stack . . . . .	134
18.2.11. Advanced - Material Scripts . . . . .	134
18.3. Circuit Property . . . . .	134
18.3.1. Volume objects . . . . .	134
18.3.2. Advanced - Surface objects . . . . .	135
18.3.3. Advanced - Line objects . . . . .	136
18.4. Thermal Property . . . . .	136
18.5. Mesh Property . . . . .	136
18.5.1. Mesh Hint . . . . .	136
18.5.2. Ignore Object . . . . .	138
18.5.3. Discretize Solid Surface only . . . . .	138
18.5.4. Discretize Solid Interior only . . . . .	138
18.5.5. Discretize Solids as wires . . . . .	138

18.6. Advanced Property . . . . .	138
18.6.1. Voxel model . . . . .	138
18.6.2. Field Storage & Display . . . . .	139
18.6.3. Measurement Definition . . . . .	140
18.6.4. Excitation Definition . . . . .	140
18.6.5. Layout Export hints . . . . .	140
18.6.6. User defined . . . . .	141
<b>19. Field Monitors</b>	<b>142</b>
19.1. EM Field . . . . .	142
19.1.1. Storage Options . . . . .	143
19.1.2. 3D Plot Options . . . . .	145
19.1.3. 3D Result Options . . . . .	149
19.2. Waveguide Port Modes . . . . .	150
19.3. EM Far Field . . . . .	152
19.3.1. Storage Options . . . . .	153
19.3.2. Processing Setup . . . . .	155
19.3.3. Mimo Correlation Setup . . . . .	158
19.3.4. Farfield Superposition . . . . .	159
19.3.5. 3D Plot Options . . . . .	159
19.3.6. 3D Result Options . . . . .	161
19.4. EM Probes . . . . .	162
19.4.1. Field Line . . . . .	162
19.4.2. Point Field Probe (Frequency Domain) . . . . .	164
19.4.3. Point Field Probe (Time Domain) . . . . .	166
19.5. Thermal Field Monitor . . . . .	166
19.5.1. Thermal Display . . . . .	166
19.5.2. Thermal Point Probe . . . . .	167
<b>20. Sources</b>	<b>169</b>
20.1. Ports . . . . .	169
20.1.1. Port Setup Wizard . . . . .	170
20.1.2. QTEM Ports . . . . .	172
20.1.3. Lumped Ports . . . . .	176
20.1.4. Feedline Ports . . . . .	180
20.1.5. TE-TM Waveguide Ports . . . . .	188
20.2. Volume Sources . . . . .	192
20.2.1. Plane Wave . . . . .	193
20.2.2. Field Source (Resizable) . . . . .	194
20.3. Thermal Source . . . . .	195
20.3.1. Thermal Source from EM loss . . . . .	196
20.3.2. Thermal Power Source . . . . .	196
<b>21. Variables</b>	<b>197</b>
21.1. Create Variables . . . . .	197
21.2. Control Variables . . . . .	199
21.3. Equations . . . . .	200
21.4. Sweep Simulation . . . . .	201
21.5. Sweep Results . . . . .	201
<b>22. Meshing and Simulation Domain</b>	<b>202</b>
22.1. Automatic Meshing . . . . .	202
22.1.1. Mesh Creation . . . . .	203
22.1.2. Simulation Domain . . . . .	203

22.1.3. General Mesh Options . . . . .	204
22.1.4. Mesh Mode and Mesh Hint . . . . .	205
22.1.5. Simulation Box . . . . .	206
22.1.6. Boundary Distance . . . . .	206
22.1.7. Advanced Mesh Setup . . . . .	206
22.1.8. Group Meshing . . . . .	208
22.1.9. Meshing Algorithm . . . . .	208
22.2. Manual Meshing . . . . .	210
22.2.1. Marking intervals . . . . .	211
22.2.2. Generic discretization . . . . .	211
22.2.3. Delete Lines . . . . .	211
22.2.4. Add lines . . . . .	211
22.2.5. Edit Lines . . . . .	212
22.2.6. Port discretization . . . . .	212
<b>23. Simulation Setup</b>	<b>213</b>
23.1. General Setup . . . . .	213
23.2. EM Setup . . . . .	215
23.2.1. Frequency . . . . .	215
23.2.2. Resonance Estimation . . . . .	216
23.2.3. Loss Calculation . . . . .	216
23.2.4. Accuracy . . . . .	217
23.2.5. Perfect Geometry Approximation (PGA) . . . . .	217
23.2.6. Time Domain Reflectometry (TDR) . . . . .	218
23.2.7. Misc . . . . .	218
23.3. Thermal . . . . .	218
23.3.1. Thermal . . . . .	219
23.3.2. Surface . . . . .	219
23.3.3. SAR/ACD . . . . .	219
23.4. Mesh . . . . .	219
23.5. Boundary . . . . .	220
23.5.1. Boundary conditions . . . . .	220
23.5.2. Boundary Distance . . . . .	221
23.6. Excitation . . . . .	221
23.6.1. Excitation Setup . . . . .	221
23.6.2. Pulse shape . . . . .	221
23.7. End Criteria . . . . .	222
23.7.1. End Criteria Setup . . . . .	223
23.8. Equations . . . . .	223
23.8.1. Symbolic Equation . . . . .	224
23.8.2. Calculation . . . . .	225
23.8.3. File Copy . . . . .	225
23.8.4. Touchstone File . . . . .	226
23.8.5. Field Superposition . . . . .	226
23.8.6. Far Field Calculation . . . . .	226
23.9. Port Setup . . . . .	227
23.10 Multi PC / On-disc Simulation Setup . . . . .	227
23.10.1 Slave Host Setup . . . . .	228
23.11 Simulation Options . . . . .	228
<b>24. Simulation</b>	<b>230</b>
24.1. Preprocessing . . . . .	230
24.2. Quick Start . . . . .	230

24.3. Simulation Control . . . . .	231
24.3.1. Simulation Log . . . . .	231
24.3.2. Simulation Plot . . . . .	232
24.4. Basic Simulation . . . . .	232
24.5. Advanced Simulation . . . . .	233
24.6. Batch processing . . . . .	235
24.7. Parameter Sweep . . . . .	236
24.8. Optimization . . . . .	238
24.8.1. Optimization Control . . . . .	238
24.8.2. Optimization Configuration . . . . .	239
24.8.3. Optimization Goal . . . . .	240
24.9. Remote Processing . . . . .	241
24.10 Post Processing . . . . .	242
<b>25. Results</b>	<b>243</b>
25.1. 2D Results set up . . . . .	243
25.1.1. Button Bar . . . . .	244
25.1.2. Graph Frame . . . . .	245
25.1.3. 2D Curve Display Setup - Plot Type . . . . .	247
25.1.4. 2D Curve Display Setup - Plot Format . . . . .	249
25.1.5. 2D Curve Display Setup - Results . . . . .	250
25.1.6. 2D Curve Display Setup - Marker . . . . .	251
25.1.7. 2D Curve Display Setup - Masks . . . . .	251
25.1.8. Variable Sweep & Curve Setup . . . . .	253
25.2. 3D Result Display . . . . .	253
25.2.1. Near field . . . . .	253
25.2.2. Far field . . . . .	253
25.3. Result Files . . . . .	254
25.3.1. File location . . . . .	254
25.3.2. File processing . . . . .	254
25.3.3. File formats . . . . .	256
<b>26. Options</b>	<b>257</b>
26.1. Editor Option Menu . . . . .	257
26.1.1. 2D Design . . . . .	257
26.1.2. 2D Design Advanced . . . . .	259
26.1.3. 3D Design . . . . .	259
26.1.4. 3D Design Advanced . . . . .	260
26.1.5. Import . . . . .	261
26.1.6. Export . . . . .	263
26.1.7. Export Advanced . . . . .	264
26.1.8. Components . . . . .	264
26.1.9. Conversion . . . . .	267
26.1.10 Design . . . . .	267
26.1.11 Dimensioning . . . . .	267
26.1.12 Preferences . . . . .	268
26.2. 2D Result Options . . . . .	268
26.3. 3D Result Options . . . . .	270
26.3.1. Basics . . . . .	270
26.3.2. Lighting . . . . .	272
<b>27. Project Wizard</b>	<b>273</b>
27.1. New Project . . . . .	273
27.2. Open Project . . . . .	273



27.3. Examples . . . . .	275
27.3.1. Example quick reference guide . . . . .	275
27.4. Templates - Transmission Lines & Waveguides . . . . .	275
27.4.1. Template quick reference guide . . . . .	276
27.4.2. Microstrip Line . . . . .	276
27.4.3. Strip Line . . . . .	277
27.4.4. Coplanar Waveguide . . . . .	277
27.4.5. Coaxial Waveguide . . . . .	278
27.4.6. Rectangular Wave Guides . . . . .	278
27.4.7. Plate Plane Wave . . . . .	278
27.4.8. Conformal . . . . .	278
27.5. Templates - Coupled Transmission Lines . . . . .	279
27.6. Templates - SMD chip components . . . . .	279
27.7. Templates - RLC . . . . .	279
27.7.1. Rectangular Inductor . . . . .	279
27.7.2. Interdigital Capacitor . . . . .	279
27.7.3. Meander Resistor . . . . .	279
27.8. Templates - Antennas . . . . .	279
27.8.1. Dipole . . . . .	279
27.8.2. Monopole . . . . .	280
27.8.3. Microstrip Antenna . . . . .	280
27.8.4. Inverted-F Antenna . . . . .	280
27.8.5. Pyramidal Horn Antenna . . . . .	280
27.8.6. Mobile phone . . . . .	281
27.8.7. BiQuad . . . . .	281
27.8.8. Patch Antenna Array . . . . .	281
27.9. Templates - Filters . . . . .	281
27.9.1. Coupled Lines . . . . .	281
27.9.2. Coupled Hairpin . . . . .	281
27.9.3. Tapped Hairpin . . . . .	282
27.10. Templates - Thermal . . . . .	282
27.11. Templates - Environments . . . . .	282
27.12. Tutorials . . . . .	282
27.12.1. Tutorial quick reference guide . . . . .	282
<b>28. Scripting</b>	<b>284</b>
28.1. User defined elements . . . . .	284
28.1.1. User defined polygons . . . . .	284
28.1.2. User defined library element . . . . .	285
28.2. Material distributions . . . . .	286
28.2.1. Material Script . . . . .	286
28.3. Scripted input, setups and environments . . . . .	288
28.3.1. Set Simulation Parameters . . . . .	288
28.3.2. Create Layer Stack . . . . .	288
28.3.3. Create Objects . . . . .	289
28.3.4. Create ports and monitors . . . . .	289
28.4. Scripted Simulations . . . . .	289
28.5. Scripted Postprocessing . . . . .	290
28.6. Supported Functions . . . . .	291
28.6.1. Functions in scripting.py . . . . .	291
28.6.2. Functions in gmf.py . . . . .	295

<b>III. Application Guide</b>	<b>303</b>
<b>29. EM Simulation</b>	<b>305</b>
29.1. ACD calculation (Averaged Current Density)	306
29.2. Active impedance calculation	306
29.3. Capacitance calculation	306
29.4. Circular Polarization (Antennas, Waveguides)	307
29.4.1. Narrow band	307
29.4.2. Broad band	307
29.5. Efficiency calculation (Antennas)	308
29.6. Far field animation	308
29.7. Far field plot	309
29.8. Far field recording	309
29.9. Gain calculation (Antennas)	310
29.10 Impedance calculation	310
29.11 Inductance calculation	311
29.12 Low frequency algorithm	311
29.13 Multiple port excitation (independent)	312
29.14 Near field animation	312
29.15 Near field plot	313
29.16 Near field recording	313
29.17 Optimization	314
29.18 Parameter Sweep	314
29.19 Phase center calculation (Antennas)	315
29.20 Plane wave excitation	315
29.21 Port creation (absorbing)	315
29.22 Port creation (concentrated)	316
29.23 Quality factor calculation (Inductance)	316
29.24 RCS calculation	317
29.25 SAR calculation	317
29.26 S-Parameter calculation	318
29.27 Touchstone file (postprocessing)	318
29.28 User equations (postprocessing)	319
29.29 Waveguide excitation (hollow)	319
<b>30. Circuit Simulation</b>	<b>321</b>
30.1. Circuit Simulation	321
30.2. EM - Circuit Co-Simulation	321
30.3. Circuit Library	323
30.3.1. General	323
30.3.2. Lumped Elements	323
30.3.3. Transmission Lines	323
30.3.4. Data Elements	324
30.3.5. Coupler	324
30.4. Circuit Simulation Setup	324
30.4.1. Circuit Simulation Configuration	324
30.4.2. Circuit Editor Options	324
<b>31. Thermal Simulation</b>	<b>326</b>
31.1. Thermal Simulation Setup	326
31.2. Thermal Solver	327

<b>32. Body Model Simulation</b>	<b>328</b>
32.1. Load Voxel File . . . . .	328
32.1.1. Import Voxel File . . . . .	329
<b>IV. Command and Setup Reference</b>	<b>330</b>
<b>33. Actions</b>	<b>331</b>
33.1. Boolean . . . . .	331
33.2. Clear . . . . .	332
33.3. Create . . . . .	332
33.4. Delete . . . . .	337
33.5. Design . . . . .	337
33.6. Display . . . . .	338
33.7. Edit . . . . .	339
33.8. File . . . . .	342
33.9. Group . . . . .	343
33.10. Mesh . . . . .	343
33.11. Misc . . . . .	344
33.12. Mode . . . . .	345
33.13. Origin . . . . .	346
33.14. Polygons . . . . .	346
33.15. Select . . . . .	347
33.16. Transform . . . . .	348
33.17. View . . . . .	351
33.18. Advanced . . . . .	352
33.18.1. Advanced/Arrows . . . . .	352
33.18.2. Advanced/Boxes . . . . .	354
33.18.3. Advanced/Create . . . . .	354
33.18.4. Advanced/Design Check . . . . .	355
33.18.5. Advanced/Edit . . . . .	358
33.18.6. Advanced/Group . . . . .	360
33.18.7. Advanced/LibraryElements . . . . .	360
33.18.8. Advanced/Linpolys & Bond Wires . . . . .	361
33.18.9. Advanced/Mesh . . . . .	362
33.18.10. Advanced/Parameters . . . . .	369
33.18.11. Advanced/Polygons . . . . .	369
33.18.12. Advanced/Properties . . . . .	372
33.18.13. Advanced/Scripting . . . . .	373
33.18.14. Advanced/Select . . . . .	373
33.18.15. Advanced/Solid . . . . .	374
33.18.16. Advanced/Transform . . . . .	374
<b>34. Editor Options</b>	<b>377</b>
34.1. 2D Design . . . . .	377
34.1.1. 2D Design/Arrow . . . . .	377
34.1.2. 2D Design/Component Snap . . . . .	377
34.1.3. 2D Design/Cursor . . . . .	378
34.1.4. 2D Design/General Snap . . . . .	379
34.1.5. 2D Design/Point . . . . .	380
34.1.6. 2D Design/View . . . . .	380
34.2. 2D Graphs . . . . .	381
34.3. 3D Design . . . . .	381

34.4. 3D Display . . . . .	382
34.4.1. 3D Display/Field Monitor Animation . . . . .	382
34.4.2. 3D Display/Lights . . . . .	382
34.4.3. 3D Display/View . . . . .	384
34.5. Components . . . . .	385
34.5.1. Components/Bond Wires . . . . .	385
34.5.2. Components/Polygons . . . . .	385
34.5.3. Components/Solids . . . . .	387
34.5.4. Components/Via Fence . . . . .	389
34.6. Design . . . . .	391
34.7. Dimensioning . . . . .	392
34.8. Export . . . . .	393
34.8.1. Export/2D Image . . . . .	393
34.8.2. Export/2D Layout . . . . .	394
34.8.3. Export/3D Image . . . . .	395
34.8.4. Export/3D Solids . . . . .	395
34.8.5. Export/Advanced . . . . .	396
34.9. Import . . . . .	398
34.9.1. Import/2D Layout . . . . .	398
34.9.2. Import/3D Solids . . . . .	400
34.9.3. Import/Empire . . . . .	401
34.9.4. Import/Advanced . . . . .	401
34.10 Preferences . . . . .	402
<b>35. Autodisc Options . . . . .</b>	<b>403</b>
35.1. General . . . . .	403
35.2. Advanced . . . . .	403
35.2.1. Advanced/Options . . . . .	403
35.2.2. Advanced/User Setup . . . . .	404
35.3. Axis Setup . . . . .	407
35.4. Simulation Box . . . . .	409
35.4.1. Simulation Box/X . . . . .	409
35.4.2. Simulation Box/Y . . . . .	409
35.4.3. Simulation Box/Z . . . . .	409
<b>36. Simulation Options . . . . .</b>	<b>411</b>
36.1. Accuracy . . . . .	411
36.2. Additional Space . . . . .	412
36.3. Advanced . . . . .	413
36.3.1. Advanced/Gabriel Object Options . . . . .	413
36.4. Boundary Conditions . . . . .	413
36.5. Frequency . . . . .	415
36.6. General . . . . .	416
36.7. Loss Calculation . . . . .	417
36.8. Meshing1 . . . . .	418
36.9. Meshing2 . . . . .	418
36.10 Port Setup . . . . .	419
36.11 Thermal . . . . .	419
36.11.1 Thermal/SAR/ACD . . . . .	420
36.11.2 Thermal/Surface . . . . .	420

<b>V. Error Messages</b>	<b>422</b>
<b>37. GUI Messages</b>	<b>424</b>
37.1. General . . . . .	424
37.2. Fields and Animation . . . . .	424
37.3. Objects . . . . .	425
37.4. Import and Export . . . . .	427
<b>38. Compilation Messages</b>	<b>429</b>
<b>39. Kernel Messages</b>	<b>431</b>
<b>VI. Theoretical Background</b>	<b>433</b>
<b>40. General Description</b>	<b>434</b>
40.1. Simulation Time . . . . .	434
40.2. Performance . . . . .	434
40.3. Number of Timesteps . . . . .	435
40.4. Number of Cells . . . . .	435
40.5. Simulation Memory . . . . .	435
40.6. Simulation Assumptions . . . . .	435
40.7. The Yee algorithm . . . . .	436
<b>41. Discretization</b>	<b>439</b>
41.1. Meshing . . . . .	439
<b>42. Boundary Conditions</b>	<b>440</b>
42.1. Electric Wall . . . . .	440
42.2. Magnetic Wall . . . . .	440
42.3. Absorbing Boundaries . . . . .	441
42.3.1. Absorbing sheet . . . . .	441
42.3.2. Perfectly Matched Layer . . . . .	441
<b>43. Stability Conditions</b>	<b>442</b>
43.1. Uniform grid . . . . .	442
43.2. Non-uniform grid . . . . .	442
43.3. Open boundaries . . . . .	442
<b>44. Excitations</b>	<b>444</b>
44.1. Types of excitation . . . . .	444
44.1.1. Concentrated source excitation . . . . .	444
44.1.2. Matched source excitation . . . . .	444
44.1.3. Plane wave excitation . . . . .	444
44.1.4. Waveguide excitation . . . . .	445
44.2. Time signals . . . . .	446
<b>45. Ports and S–matrices</b>	<b>447</b>
45.1. Port definition . . . . .	447
45.1.1. Quasi-TEM ports . . . . .	448
45.1.2. Lumped ports . . . . .	449
45.1.3. Waveguide ports . . . . .	449
45.2. Time signals . . . . .	450
45.2.1. Absorbing Port signal processing . . . . .	451
45.2.2. Concentrated Port signal processing . . . . .	451

45.3. DFT . . . . .	451
45.4. Impedances . . . . .	452
45.5. Wave quantities . . . . .	452
45.6. Scattering parameters . . . . .	452
<b>46. Specials</b>	<b>453</b>
46.1. Y-Parameter Processing . . . . .	453
46.2. Hollow Waveguides . . . . .	453
46.3. Near to Far Field Transformation . . . . .	454
46.4. SAR and ACD . . . . .	455
46.5. Low-Frequency Algorithm . . . . .	455
46.6. Dielectric Properties of Biological Tissue . . . . .	456
<b>VII.Literature</b>	<b>457</b>
<b>Bibliography</b>	<b>458</b>
<b>VIII Index</b>	<b>459</b>
<b>Reference index</b>	<b>460</b>
List of Actions . . . . .	460
List of Options . . . . .	462
List of Autodisc Options . . . . .	463
List of Simulation Options . . . . .	463

# User and Reference Manual

for the  
3D-EM Time Domain Simulator  
EMPIRE XPU™  
Version 9.0

©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](mailto:empire.support@imst.de) • Internet: [www.empire.de](http://www.empire.de)

## Copyright notice

Copyright ©1998-2024 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 – 2024. 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 Electro Magnetic field simulator for the analysis of Packages, Interconnects, Radiators, waveguide Elements 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 up to 50000 million cells per seconds (MCells/s) on a single PC 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.

## 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.

- **Part I** describes the installation and licensing process for the different platforms.
- **Part II** explains the EMPIRE XPU™ Graphical User Interface, the simulation control as well as pre- and post-processing features.
- **Part III** is a quick guide for typical applications with references to the details given in the later chapters and sections.
- **Part IV** is the command and setup reference, a complete description of all actions, operations, options and simulation parameters of EMPIRE XPU™ .
- **Part V** gives a summary of warning and error messages along with comments to solve the problems.
- **Part VI** describes the theoretical background of the applied method.
- **Part 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](mailto: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 i7/9, Xeon, AMD Ryzen, Threadripper, Epyc ) with at least 4 GB of RAM
- Microsoft Windows 10/11 or Windows Server on 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

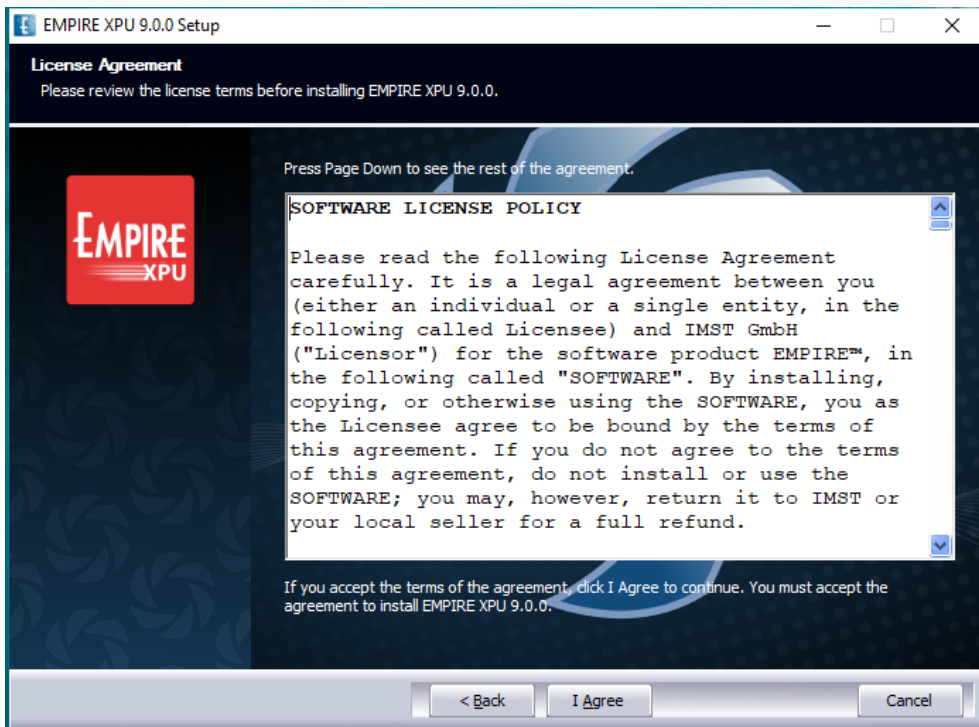


Figure 1.2.: EMPIRE XPU™ License Agreement Window

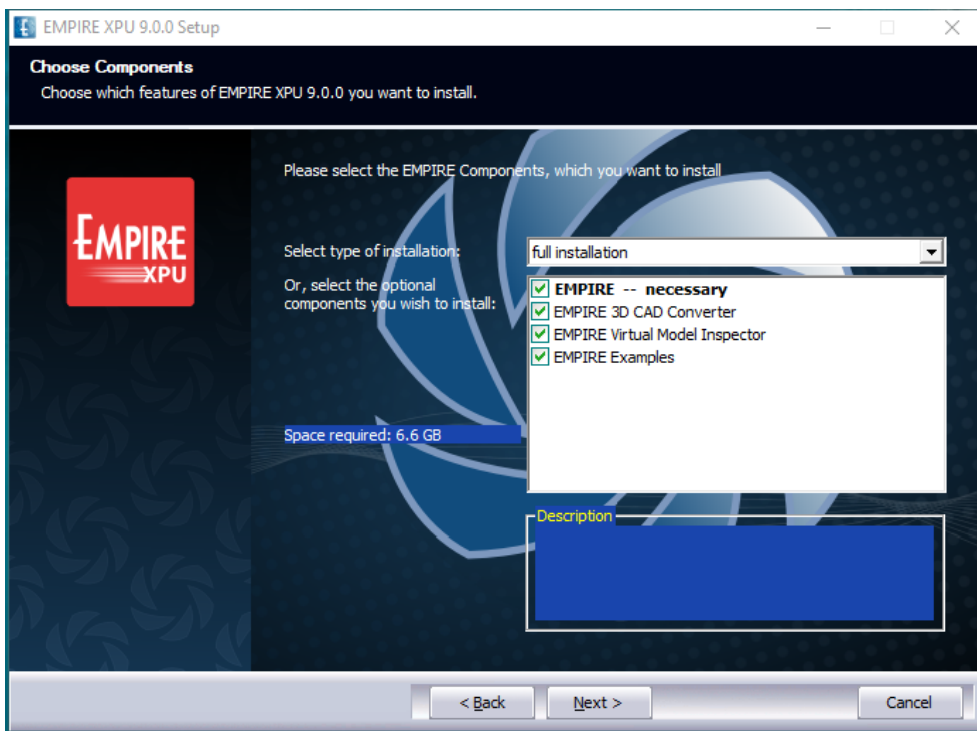


Figure 1.3.: EMPIRE XPU™ Choose Components Window

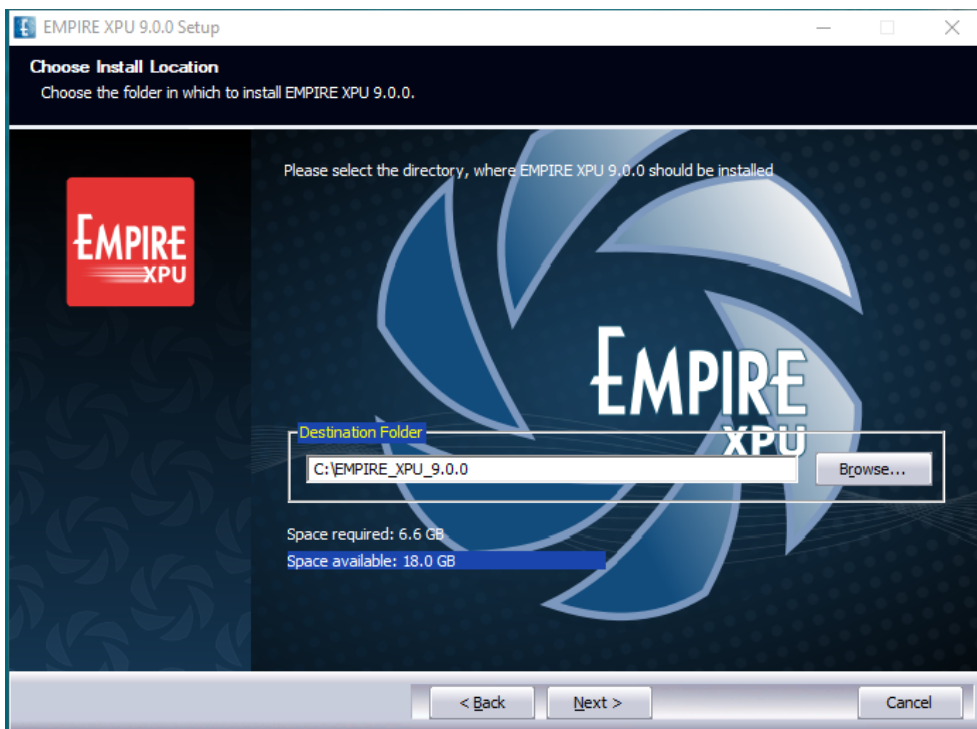


Figure 1.4.: EMPIRE XPU™ Choose Install Location Window

- Step 5:**
- Please confirm or browse the **Installation Folder**<sup>a</sup>. Do **not** use folders with spaces such as **Program Files**<sup>b</sup>
  - Check the space required and if your system has enough memory available
  - Press **Next>** to continue the installation

<sup>a</sup>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

<sup>b</sup>Note, on some systems spaces are not allowed in installation path.

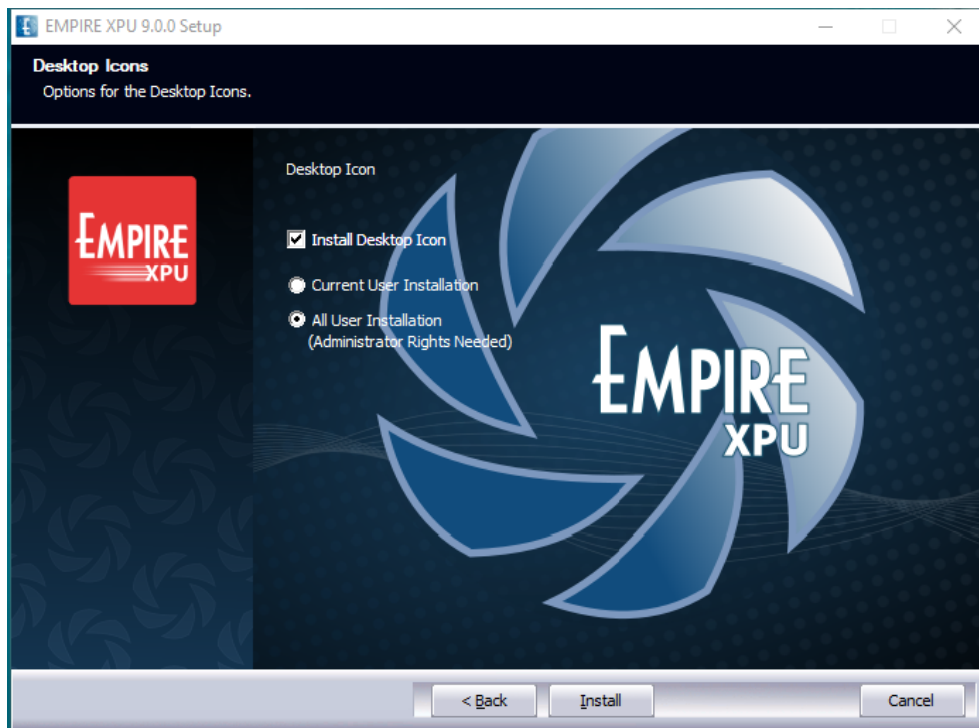


Figure 1.5.: EMPIRE XPU™ Install Desktop Icon Window

- 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

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_9.0.0\`
- 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 obtained with the following methods

- **License Request** Starting Empire and clicking the **Request** button, see Fig.2.3
- **License Management Tool** Starting `lmttools.exe` in `C:\EMPIRE_XPU_9.xx\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`

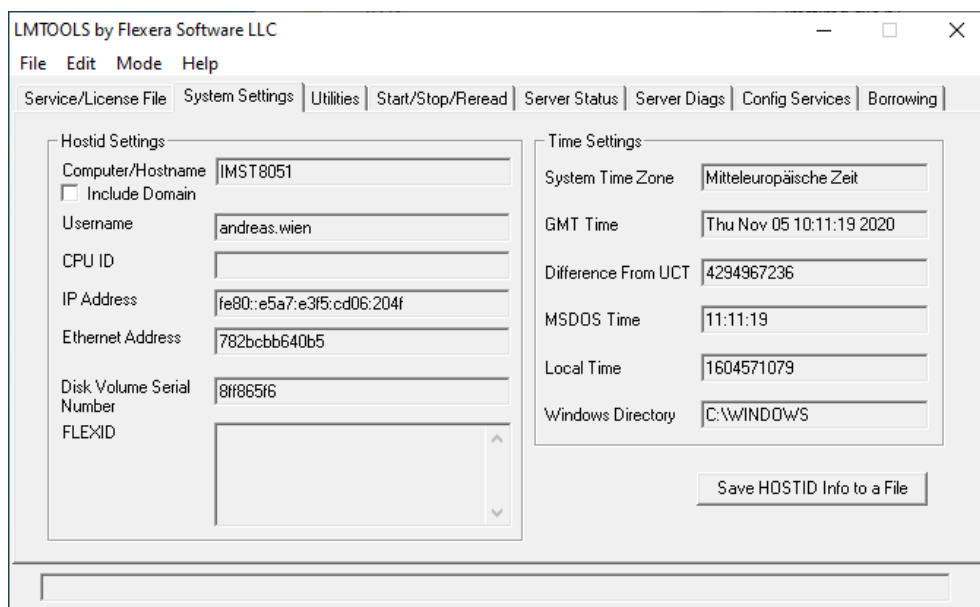


Figure 2.1.: License Management Tool

## 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/Serial port 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.xx\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

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

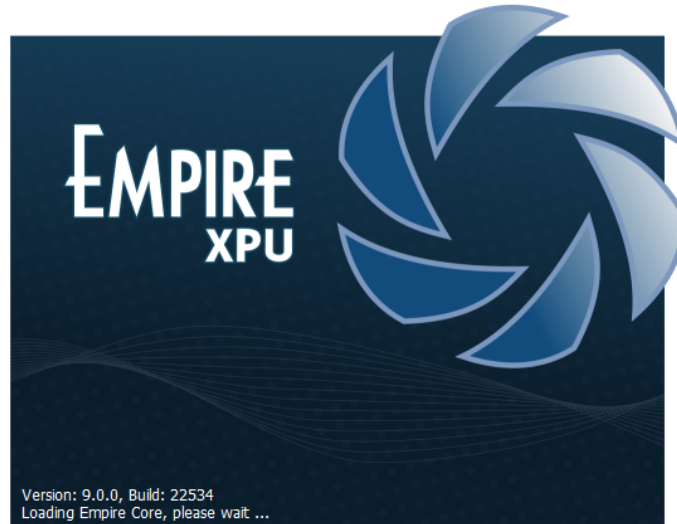


Figure 2.2.: EMPIRE XPU™ Start-up screen

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

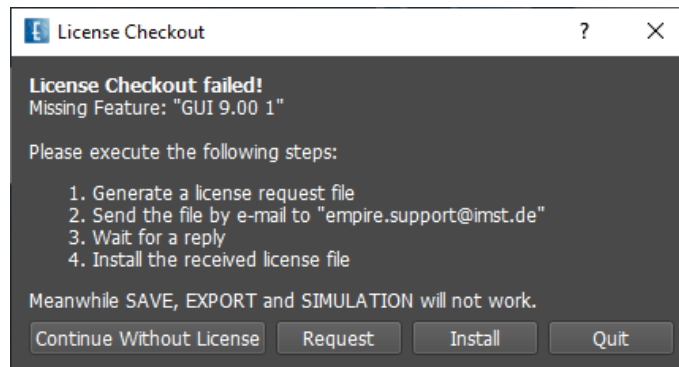


Figure 2.3.: EMPIRE XPU™ License check

**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.

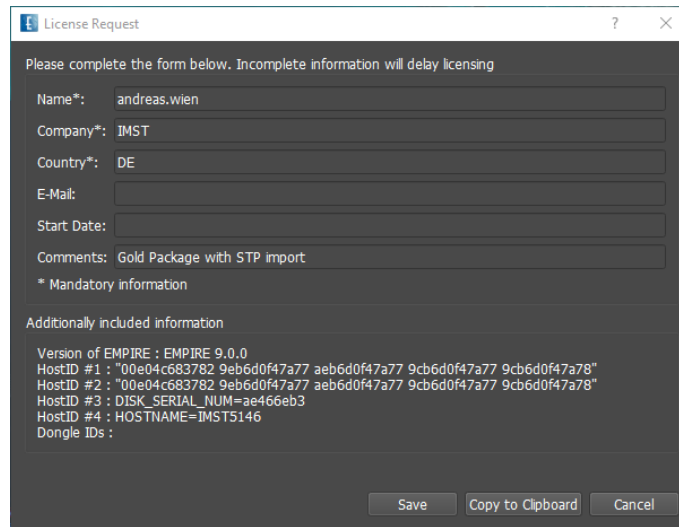


Figure 2.4.: EMPIRE XPU™ License Request Window

### 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 7/10 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.xx\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:

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

- Double click the desktop icon
- Double click an existing file with the ending `.emx` (or `.gym` for compatibility reasons)
- Start Menu - All Programs - EMPIRE XPU™ 9.x.x - empire-9.x.x
- a command entered in the command window (no line break):  

```
start "" "C:\EMPIRE_XPU_9.0.0\empire\PY3k-W64\Python3k\python.exe" -0 -u  
"C:\EMPIRE_XPU_9.0.0\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: 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:

```
"C:\EMPIRE_XPU_9.xx\empire\PY3k-W64\Python3k\python.exe" -0 -u  
"C:\EMPIRE_XPU_9.xx\empire\env.py" simul msl_line.emx  
for starting the simulation.
```

#### 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

### 3.1. Simple Test Procedure

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

1. Double click Desktop icon
  - Check: if the full screen comes up, see Figure 3.1, the license is working
2. Select the **Templates** tab and click OK.

### 3. Starting Empire on Windows

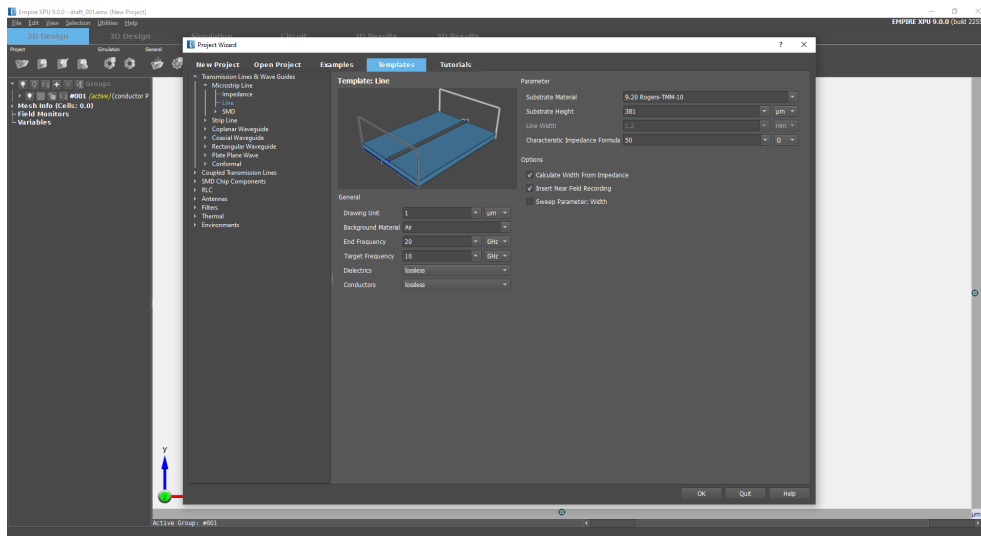


Figure 3.1.: EMPIRE XPU™ Start-up Window

- Check: If the MSL line example is visible, see Figure 3.2, the OpenGL graphic is working

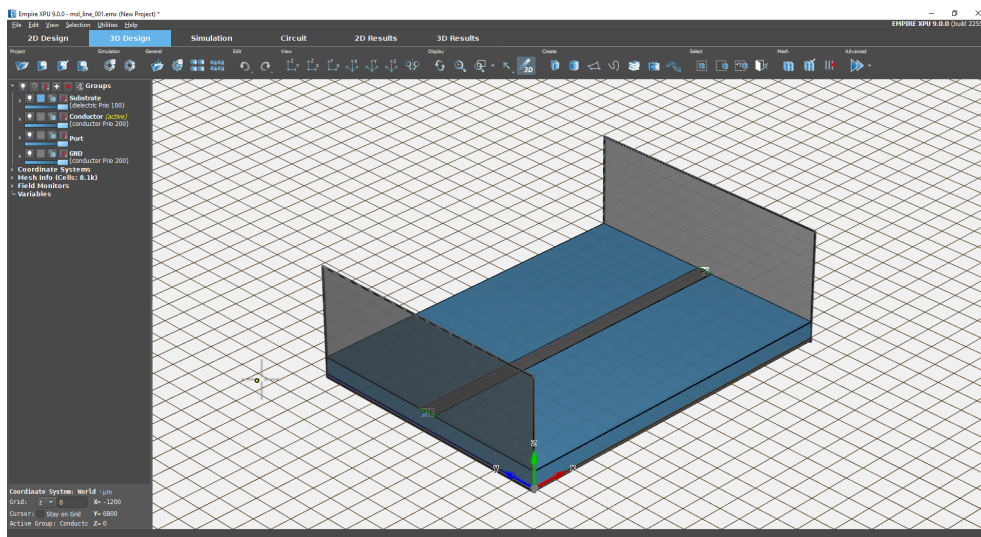



Figure 3.2.: EMPIRE XPU™ Standard Template

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 simulation kernel is working
4. Click Tab "2D Results",
  - Check: If the S-Parameter results are plotted as in Figure 3.4 the simulation was successful
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 Visualization Tool Kit (VTK) works

### 3. Starting Empire on Windows

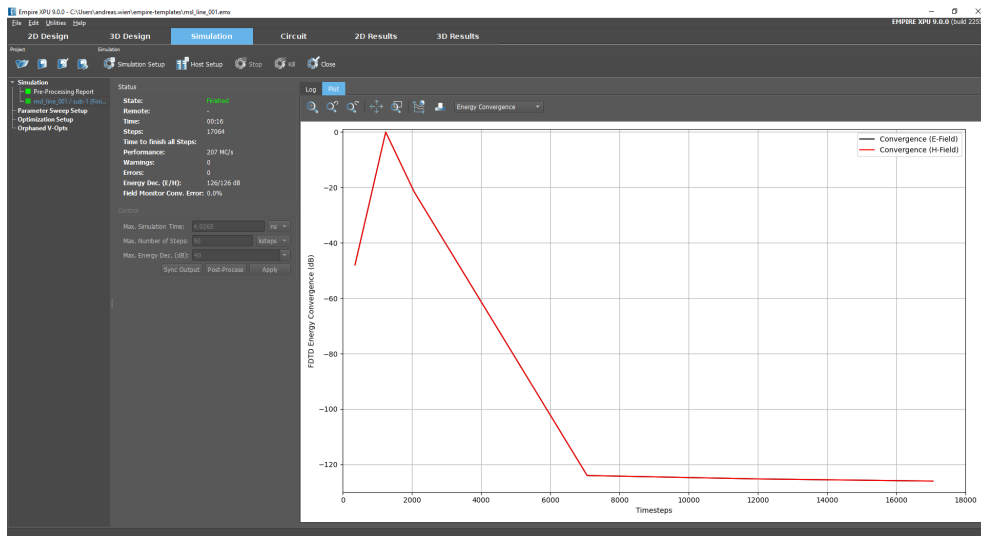


Figure 3.3.: EMPIRE XPU™ Standard Template simulation convergence

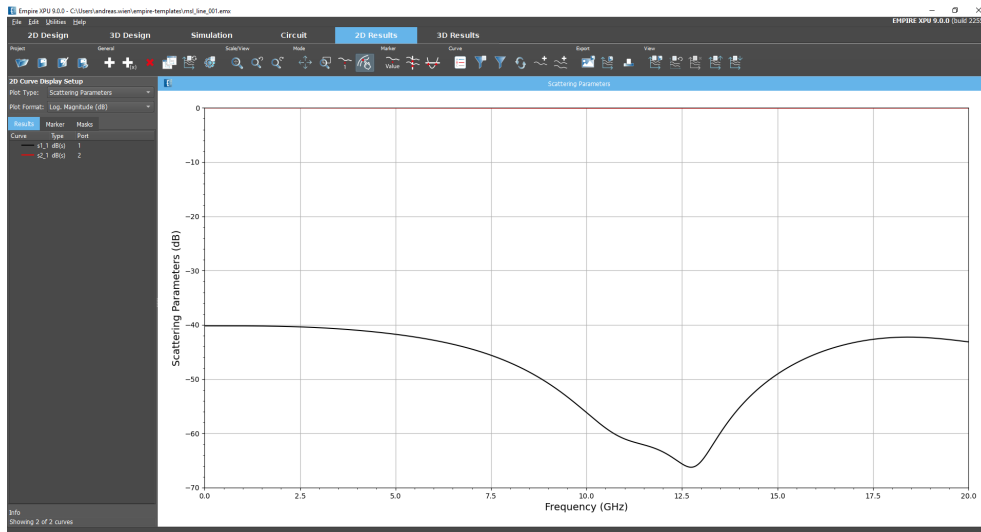


Figure 3.4.: EMPIRE XPU™ 2D Result Plot



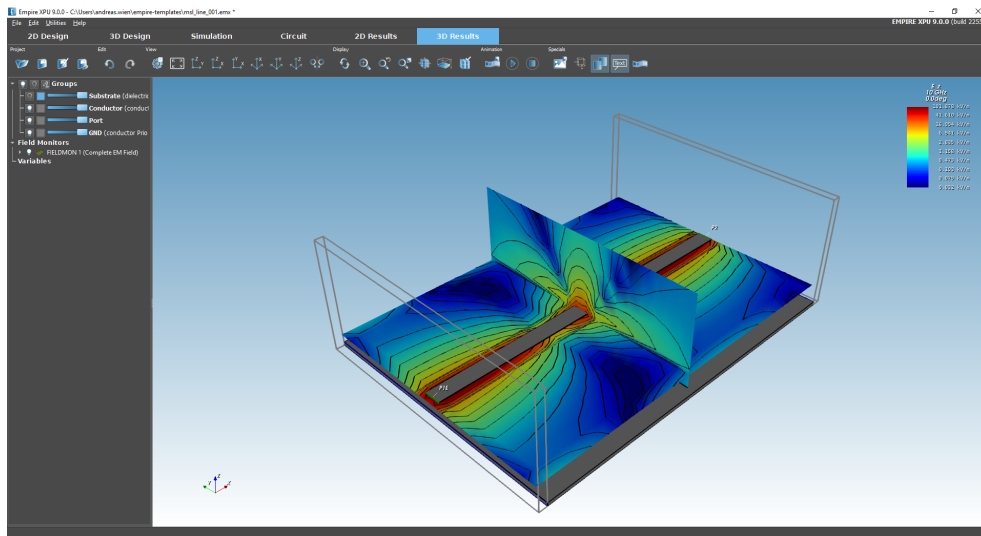


Figure 3.5.: EMPIRE XPU™ 3D Result Plot

## 3.2. 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

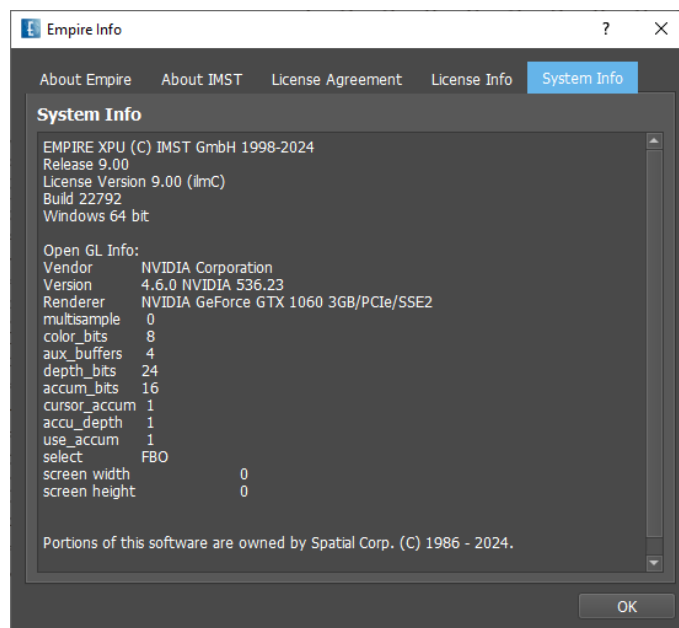


Figure 3.6.: EMPIRE XPU™ System Info

## 4. Floating License on Windows

This chapter describes the installation of floating licenses. With this option, a designated PC acts as a **license server** from which all clients in the network can check out licenses if available. 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. 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
<code>SERVER hostname id# 1700</code>	<code>SERVER mylicserv id# 55005</code>
<code>DAEMON imstlmd</code>	<code>DAEMON imstlmd port=55006</code>
<code>FEATURE empire-fs imstlmd ver# date lic# ...</code>	<code>FEATURE empire-fs imstlmd ver# date lic# ...</code>
<code>...</code>	<code>...</code>

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.xx` 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 program "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.

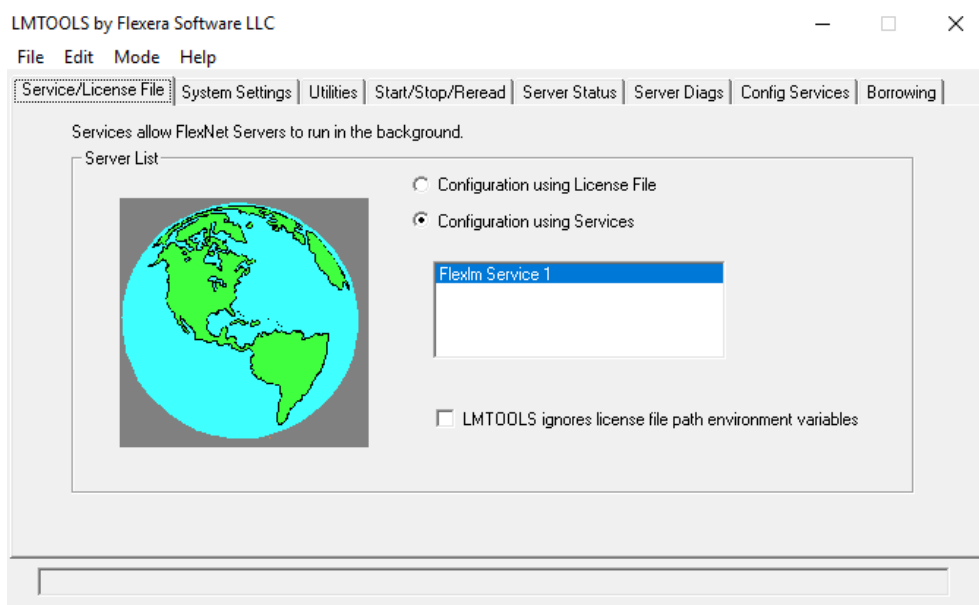


Figure 4.1.: EMPIRE XPU™ Lmtools Starting Window

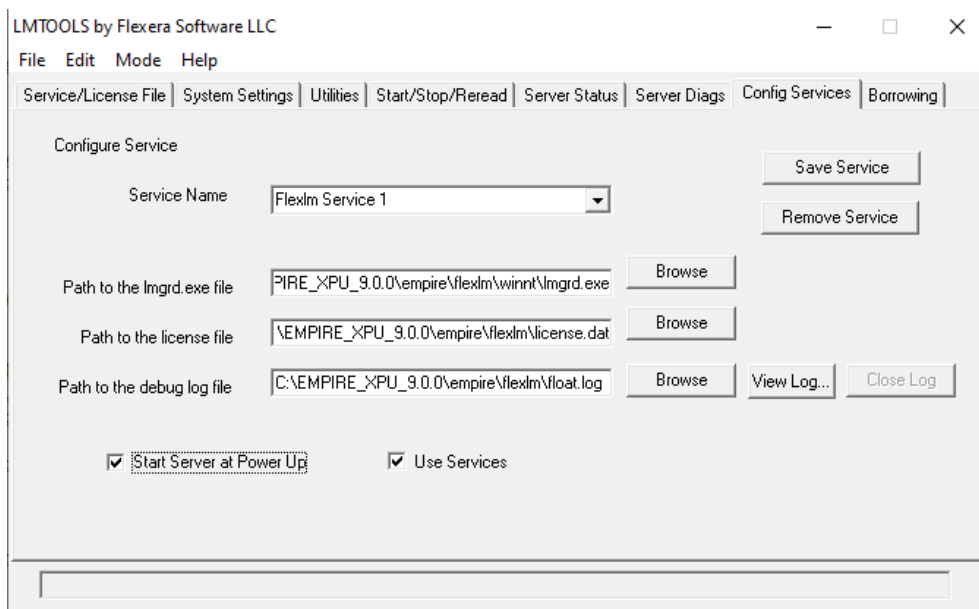


Figure 4.2.: EMPIRE XPU™ Configure Services Tab

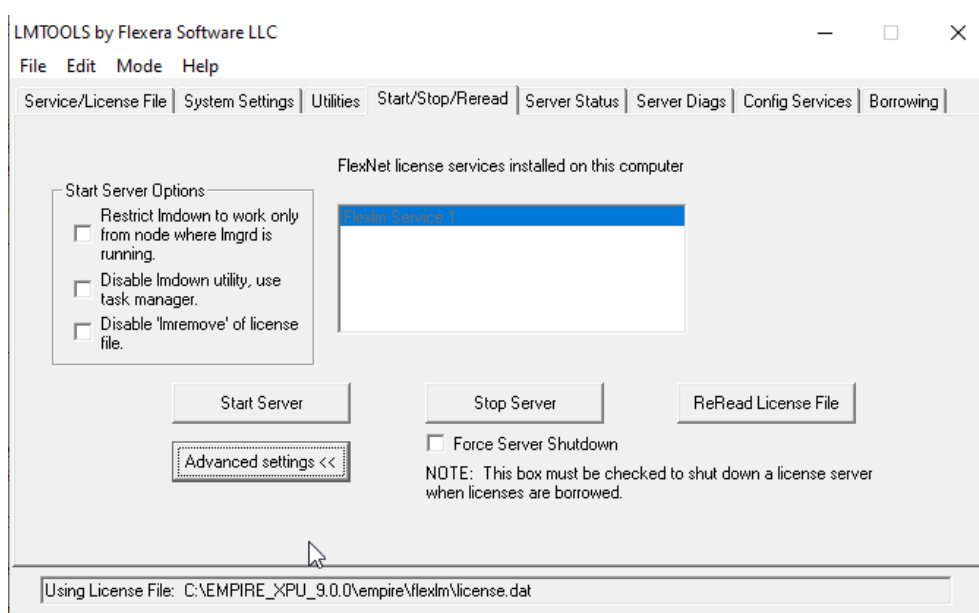


Figure 4.3.: EMPIRE XPU™ Lmtools Start Server Tab

### 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.

As an alternative EMPIRE XPU™ can be started with an option `-no1m` 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.

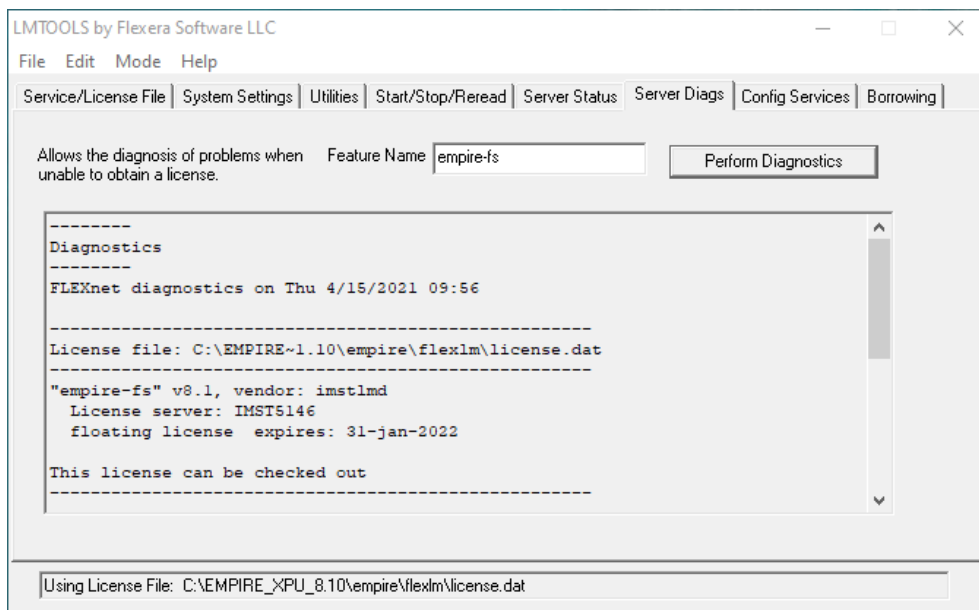


Figure 4.4.: EMPIRE XPU™ Lmtools Server Diagnostic

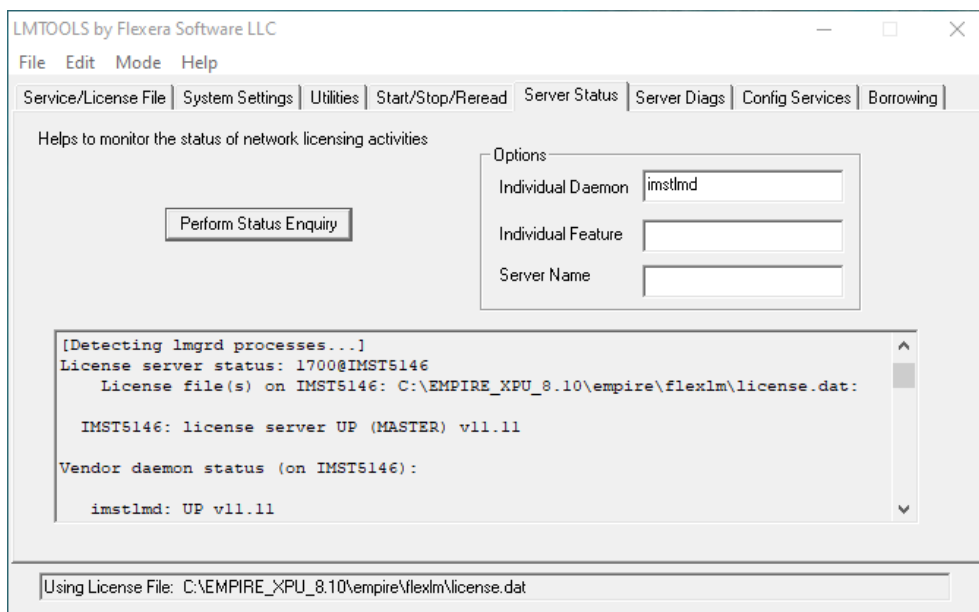


Figure 4.5.: EMPIRE XPU™ Lmtools Server Status



## 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.

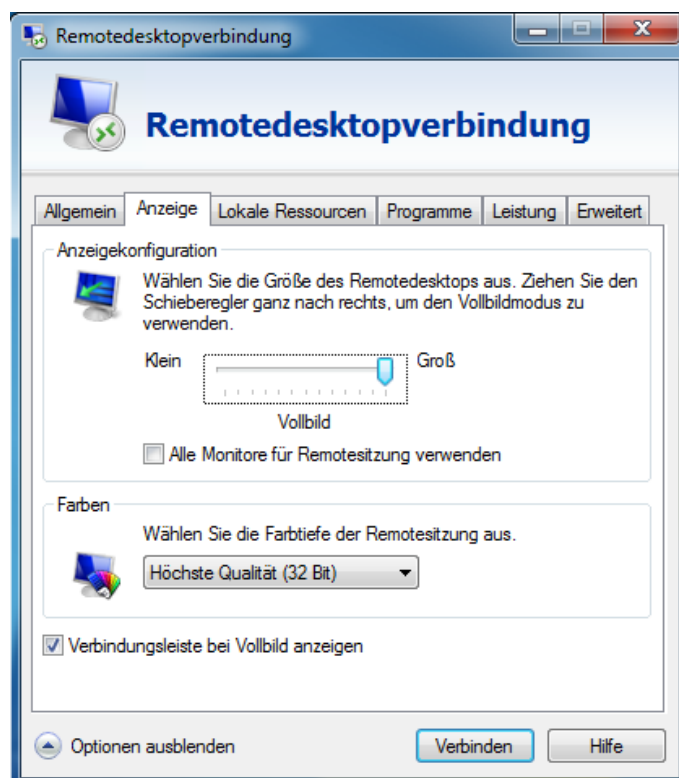


Figure 5.1.: Configuration for Windows Remote Desktop Connection

- 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 Digi**. 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 (EMPIRE-SERVER) has to be executed:

### Start Service

- Service started within EMPIRE XPU™ in Utilities - Server - Start
- Start associated Windows Service
- From Command Line:
  - Windows: `EMPIRE-SERVER.cmd` (located in `C:\EMPIRE_XPU_9.0.0\empire\binW`)
  - Linux: `./EMPIRE-SERVER`

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 addresses which is described in section 5.2.4.

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.

During the simulation the control and remote PC require a stable network connection and may not be disconnected.

## Manually Stop Service

- Service stopped within EMPIRE XPU™ in Utilities - Server - Stop
- Stop associated Windows Service
- From Command Line:
  - Windows: `EMPIRE-KILL-SERVER.cmd` (located in `C:\EMPIRE_XPU_9.0.0\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 (Legacy) - Start**, see Figure 5.2

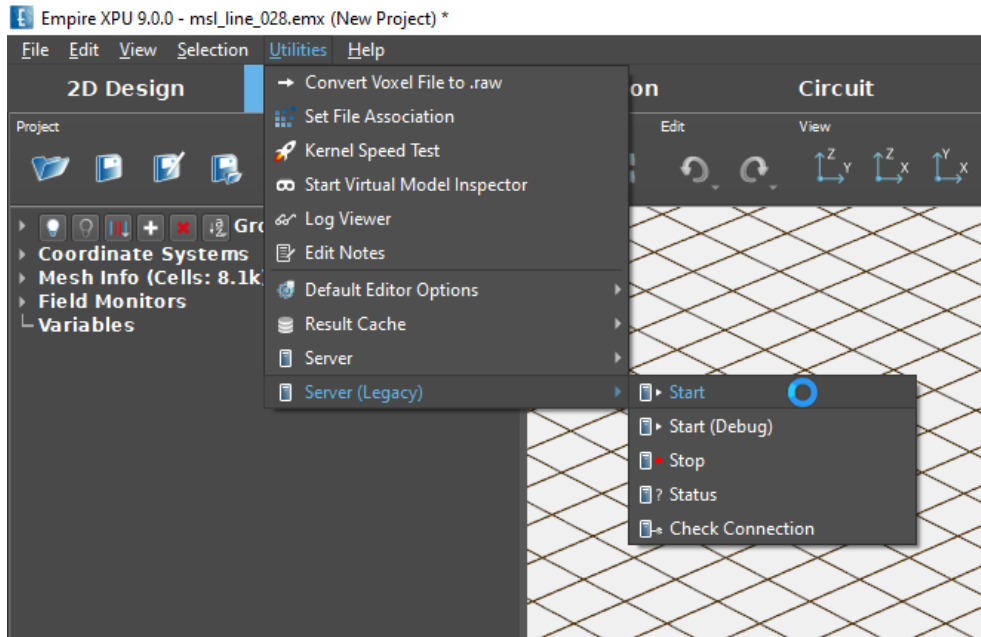


Figure 5.2.: Starting the EMPIRE XPU™ remote service from GUI

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. Starting the remote service as a Windows command

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\_9.0.0\empire\binW**

- **EMPIRE-SERVER.cmd**
- **EMPIRE-KILL-SERVER.cmd**

### 5.2.3. Starting the remote service as a Windows service

In order to start an 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**

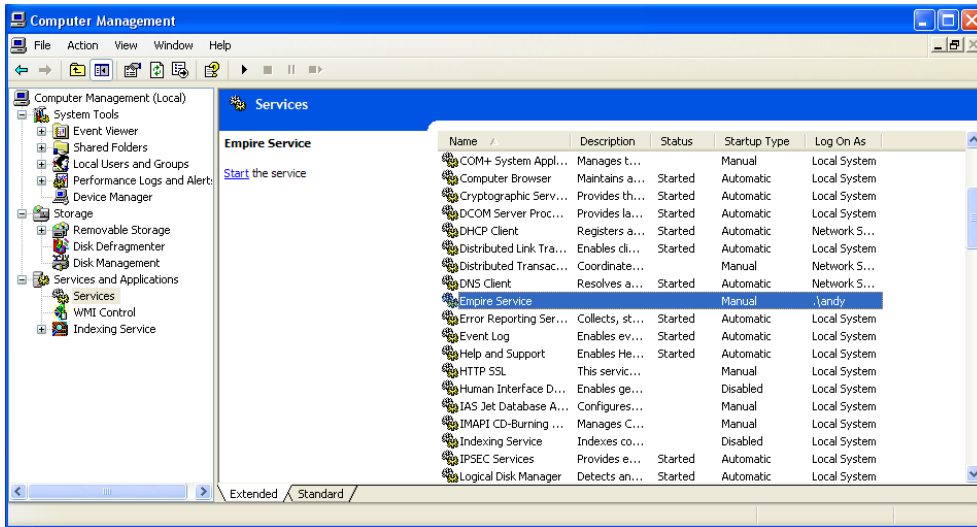


Figure 5.3.: Starting the remote server as a Windows service

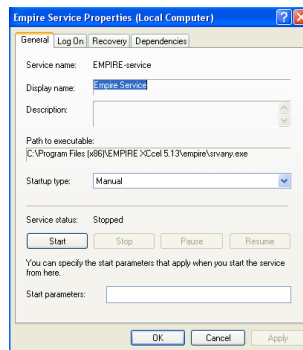


Figure 5.4.: EMPIRE XPU™ Service Properties

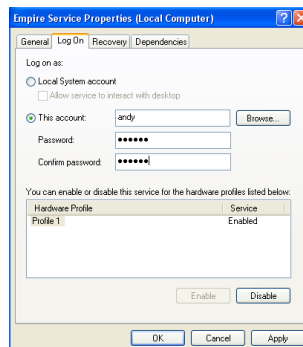


Figure 5.5.: Creating an account

#### 5.2.4. 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 - 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

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<sup>1</sup>)

### 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
- EMPIRE XPU™ 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)

---

<sup>1</sup>10 Gbit/s recommended

- 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 higher RAM. 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 (typically 2/3 of the single PC algorithm). Further, the performance depends on the network speed, simulation size and number of slaves.

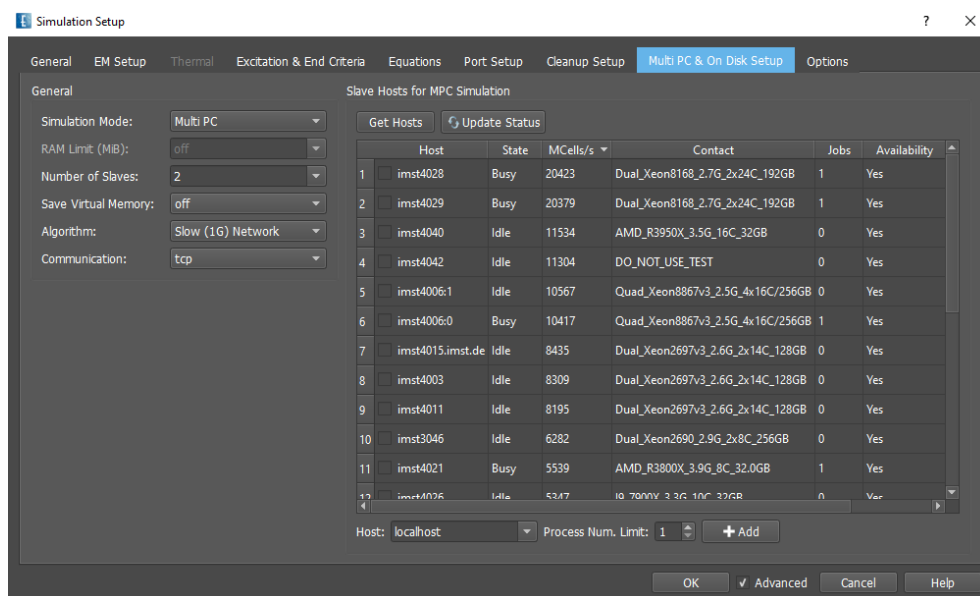


Figure 6.1.: EMPIRE XPU™ Multi-PC set up



## 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](http://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.

## 8.3. Start Commands

The following commands are available in the installation path, e.g. `~/EMPIRE_XPU_8.xx`. 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**
  - The Empire Service will be started and can now be found or added to the host list of clients. It can be 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)
- **EMPIRE-KILL-SERVER** stops the service

### 8.3.1. 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
```

## 8.4. License Request

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

### Hint: Linux Network card 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`

## 8.5. 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`

## 8.6. Floating license

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

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

### 8.6.1. 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.

## 8.7. Trouble Shooting

### 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.

### 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 multiple 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 structure 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 ftdt.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.
- **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 obtained 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)

#### Distributed Computing

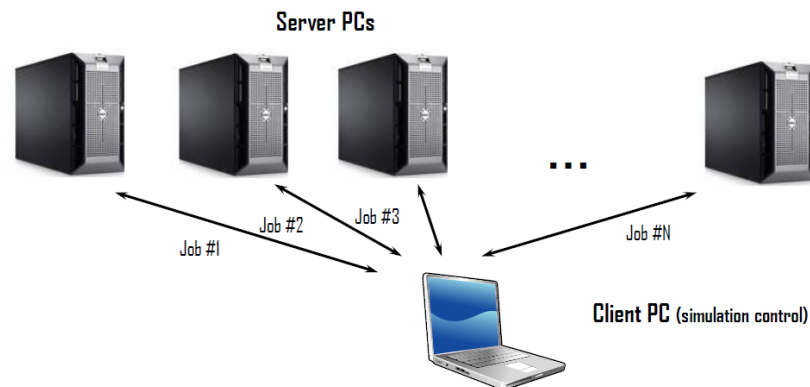


Figure 10.1.: Job distribution (using several Single-PC Solver licenses)

- 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 heterogen
- 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
- Ring-like data exchange

## Cluster Computing

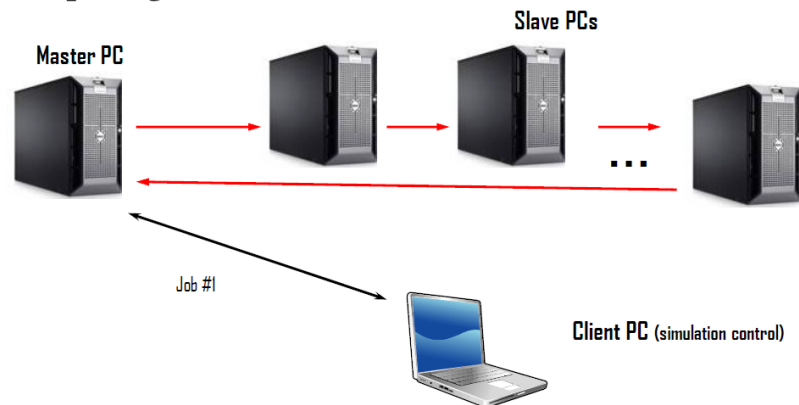


Figure 10.2.: Job sharing (using one Multi-PC Solver license)

- 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

# 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. **B12392.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. 8.00: Version 8.0, 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](mailto: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](mailto: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.com`
- China : `empire.support.cn@imst.com`
- India : `empire.support.in@imst.com`
- Germany : `empire.support@imst.com`

**Part II.**

**Empire User Interface**

## 12. Introduction

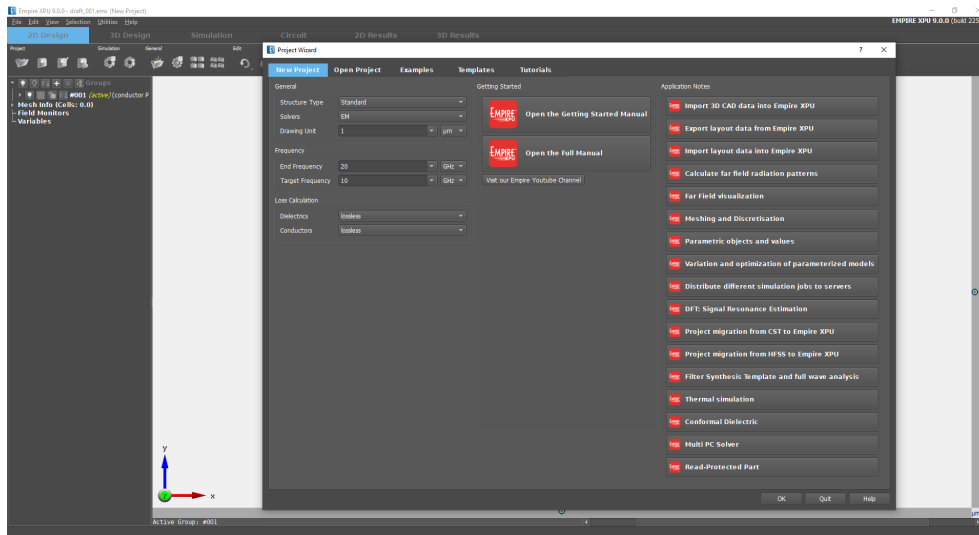


Figure 12.1.: EMPIRE XPU™ start-up window

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

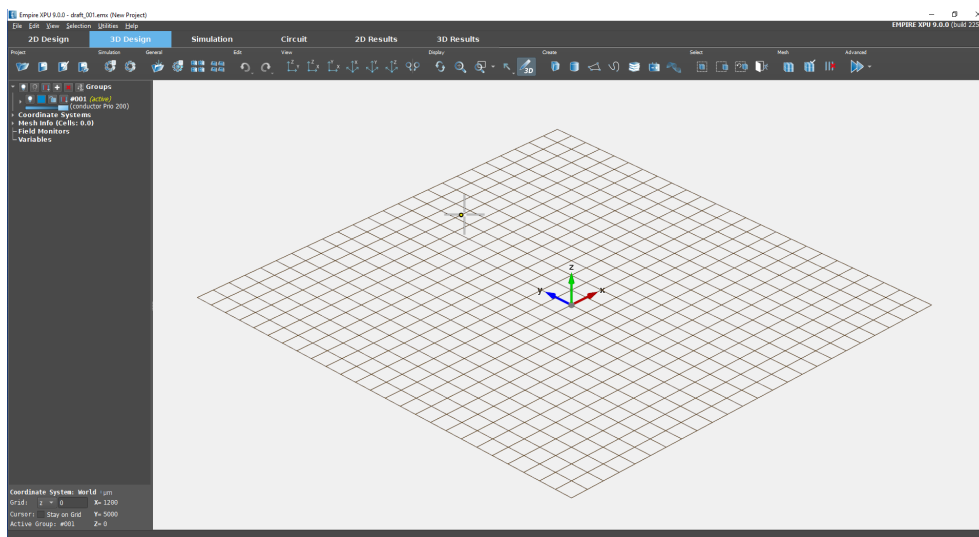


Figure 12.2.: EMPIRE XPU™ started from scratch

## 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.

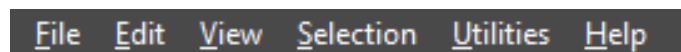


Figure 12.3.: EMPIRE XPU™ menu tool bar

## 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

- **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** of the server on the current machine
  - **Stop** of 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)** Start 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
- **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

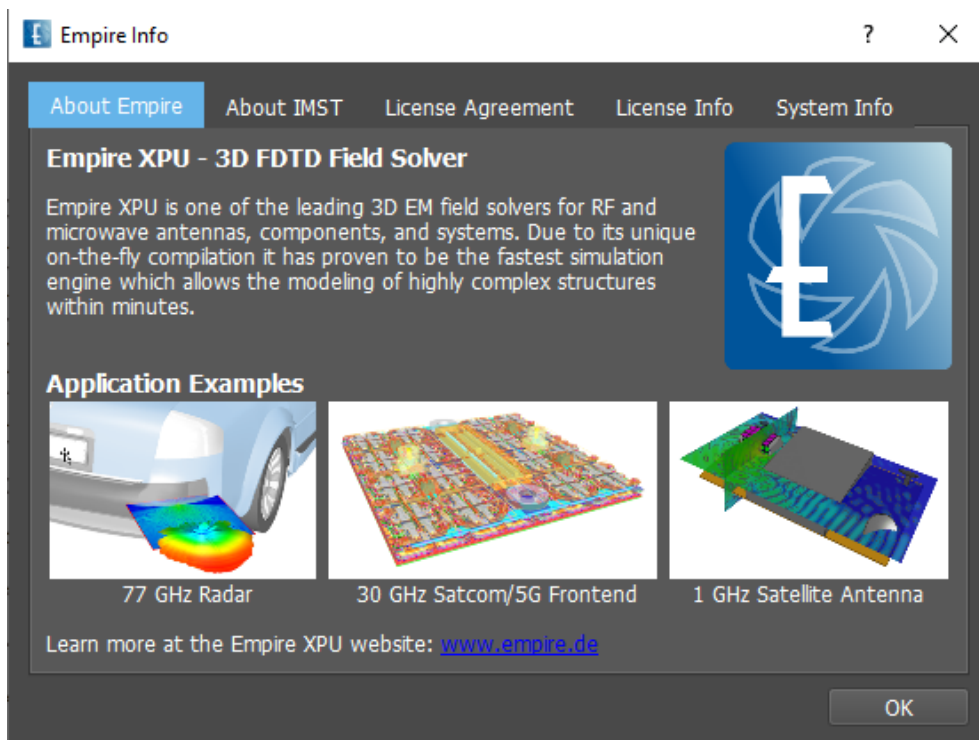


Figure 12.4.: EMPIRE XPU™ Help - Info window



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. 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  $x=\text{const}$ ,  $y=\text{const}$ , and  $z=\text{const}$  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 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

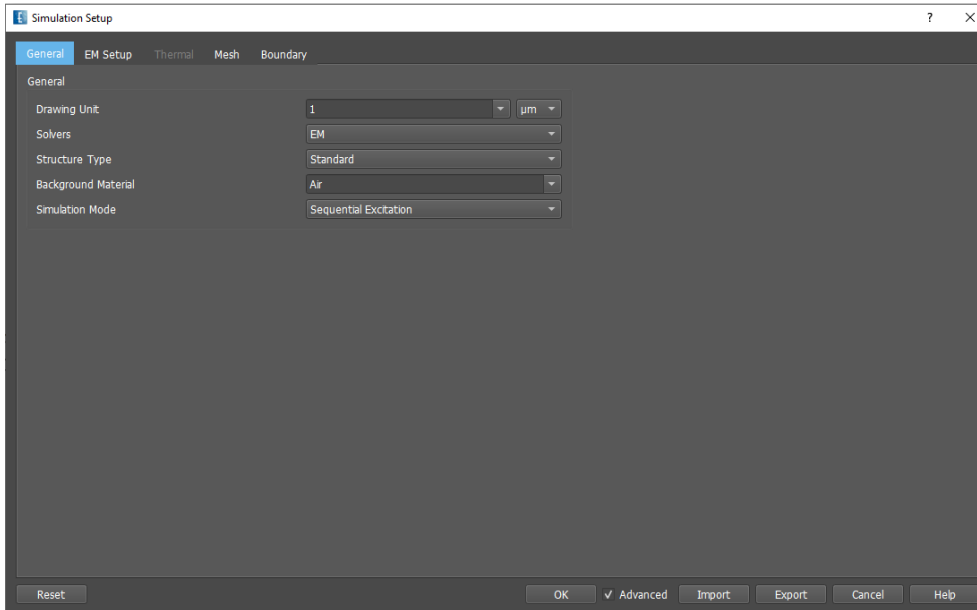


Figure 12.7.: EMPIRE XPU™ Simulation Settings

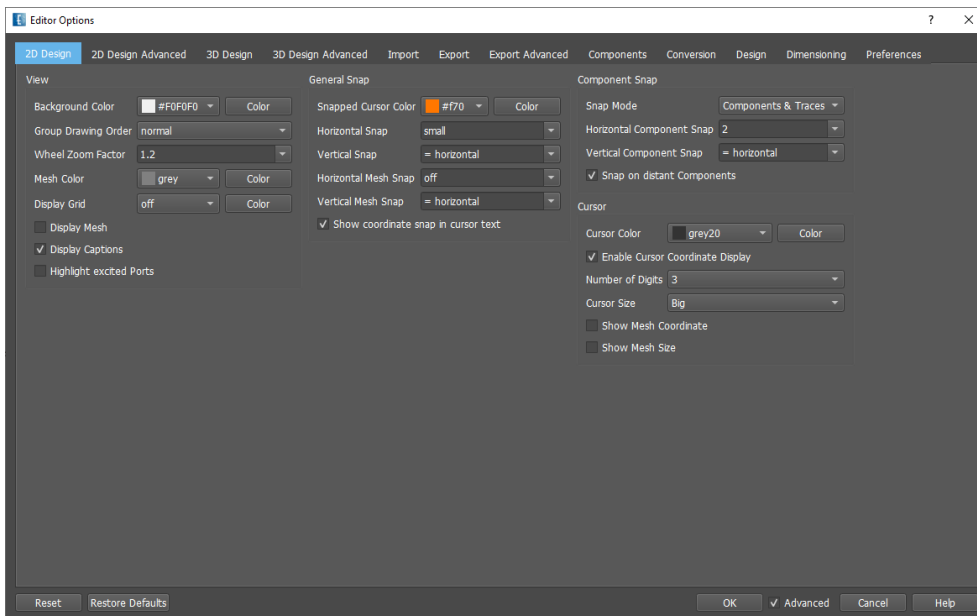


Figure 12.8.: EMPIRE XPU™ Editor Options

## 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.  $x=\text{const}$ ,  $y=\text{const}$ , or  $z=\text{const}$
- 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  $x=x_0$ ,  $y=y_0$ ,  $z=z_0$ , move the cursor to  $x_0,y_0$ , press the space key, and move the cursor to  $y_0$ ,  $z_0$ .

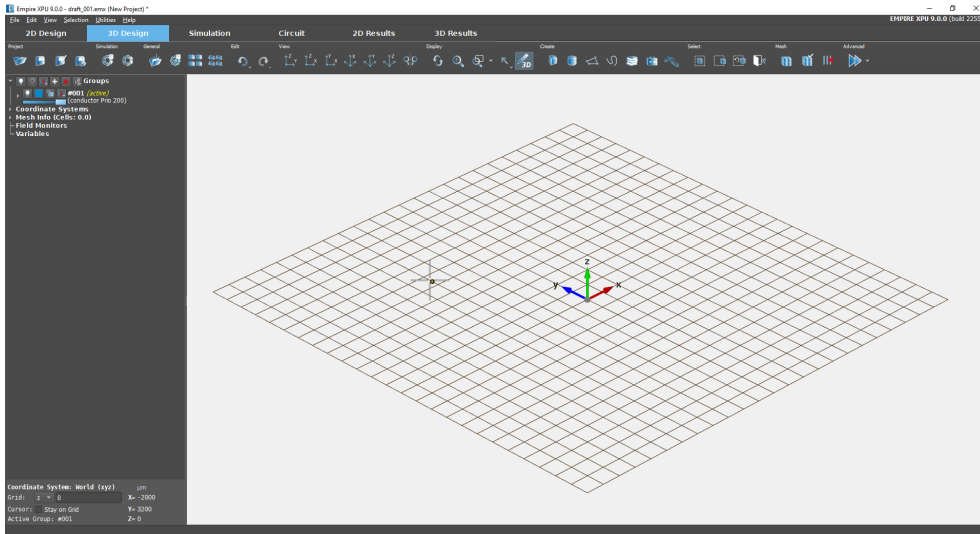


Figure 12.9.: 3D Design mode display showing tripod and xy grid lines

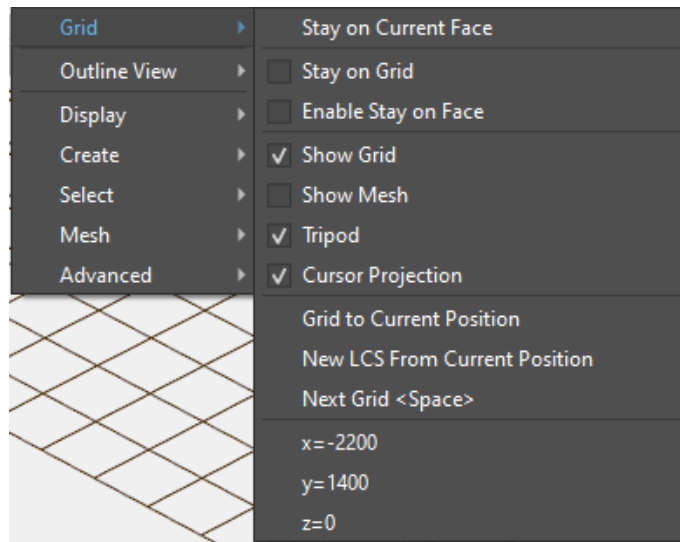










Figure 12.10.: EMPIRE XPU™ Context Menu (right click in workspace)

### 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 left 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 left click an **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 entering a point and applying the function  **Change origin of Word Coordinate System**
- The WCS is displayed with a tripod labeled 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 - Coordinate 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 - Coordinate 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.

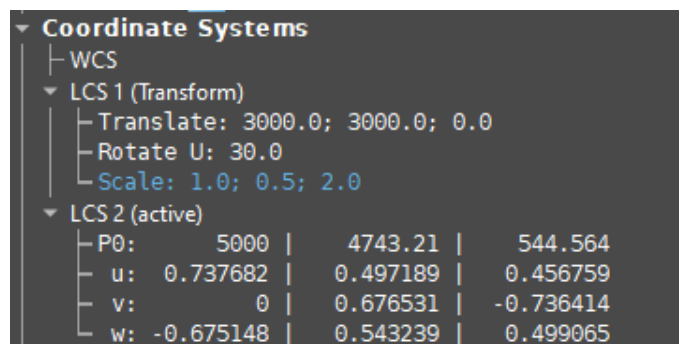


Figure 12.11.: Example of Coordinate Systems


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 extents** 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  $\uparrow\leftarrow\downarrow\rightarrow$  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




## 12.3. 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  $\uparrow\leftarrow\downarrow\rightarrow$  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.

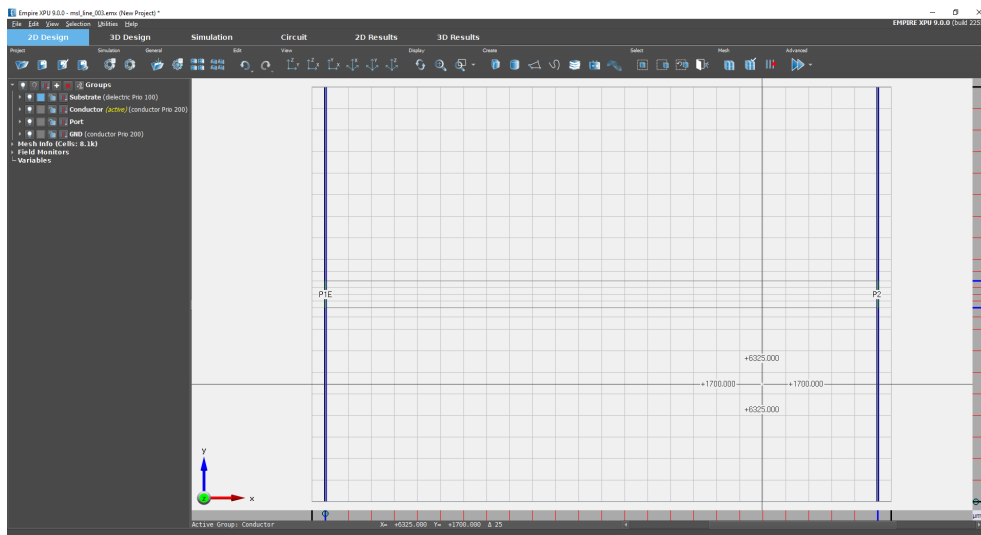



Figure 12.12.: 2D Design mode display showing mesh lines

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**

### 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** 

## 12.4. 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** (left mouse button) a continuous zoom is possible:
  - **zoom fine**: drag horizontally
  - **zoom coarse**: drag vertically

### 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-through 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

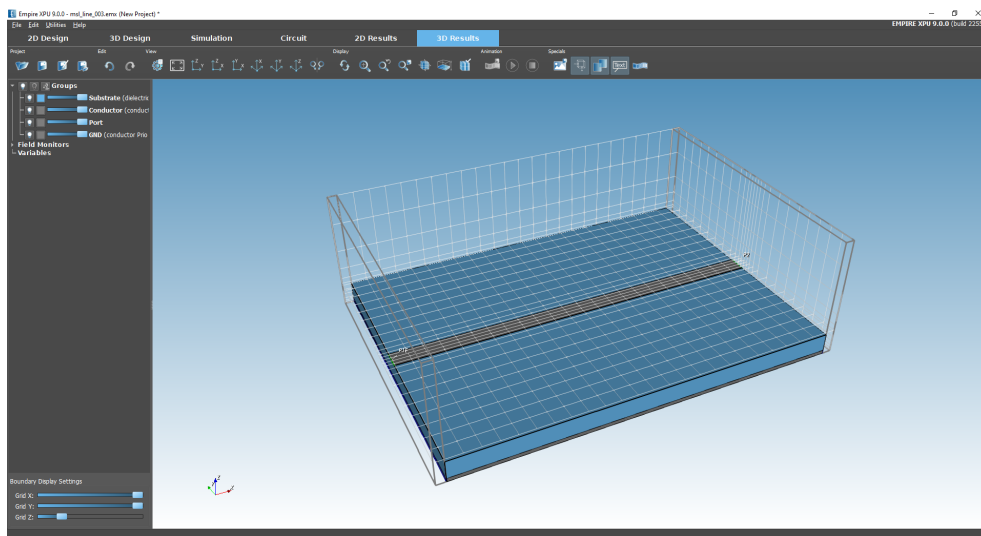


Figure 12.13.: 3D Results mode showing mesh lines

### 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, .vrm, .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 indicated in 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

**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



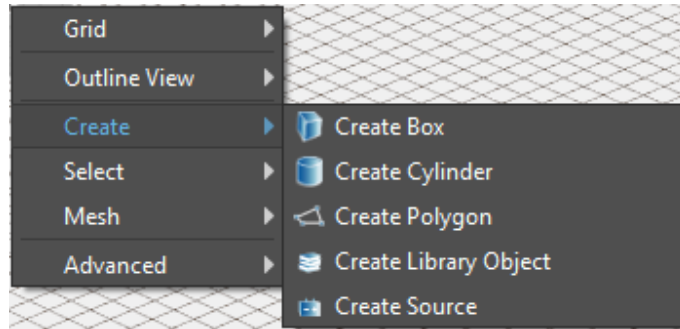


Figure 12.15.: Context menu (right mouse click)

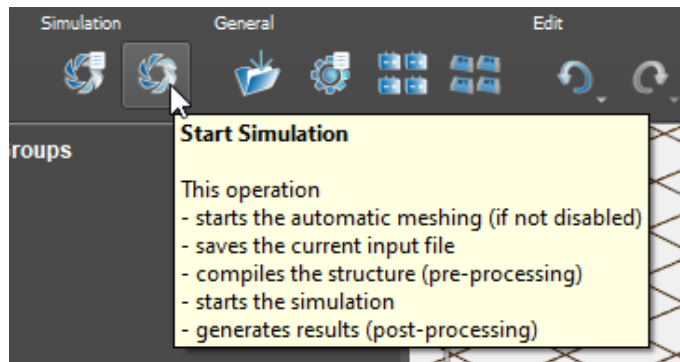


Figure 12.16.: Help on item (hover pointer over icon or input field)

## 12.6. Side menu / Navigation tree

On the left side of the window there are areas for

- the side menu
- grid and coordinates
- 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 ▼.

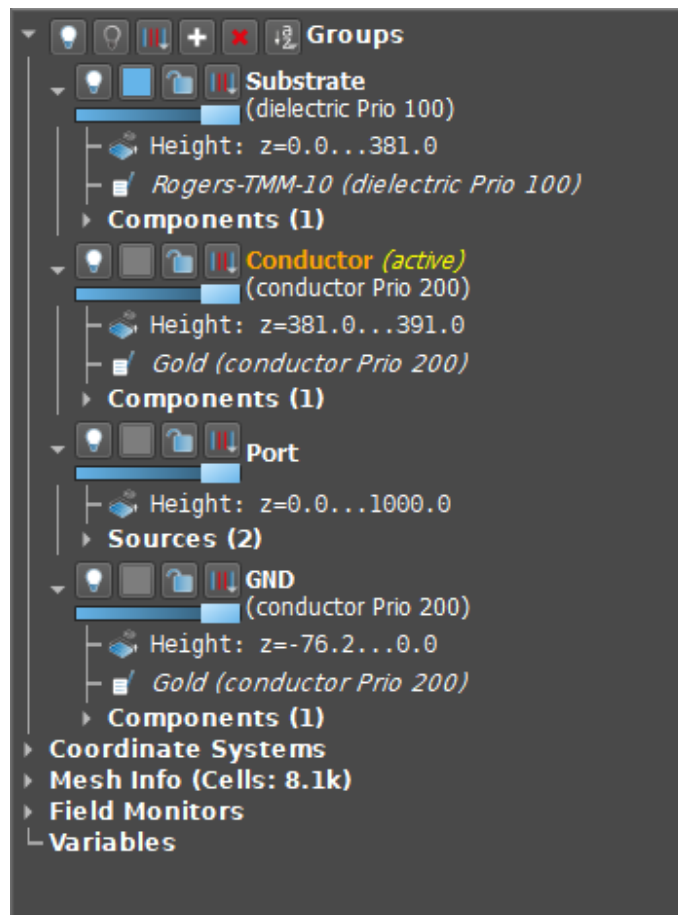


Figure 12.17.: EMPIRE XPU™ navigation tree

- **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.

## 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:

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

After selection:

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

After arrow input:

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

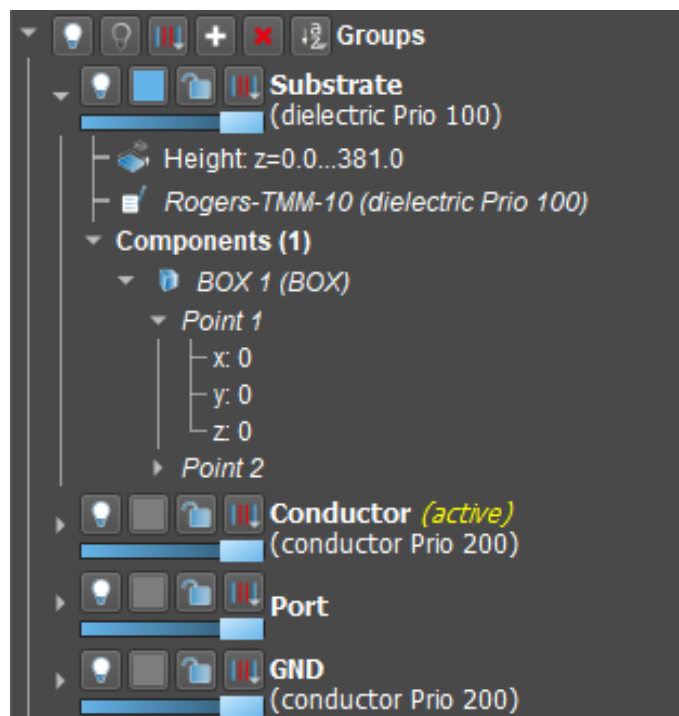






















Figure 14.1.: Group menu

## 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
-  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.



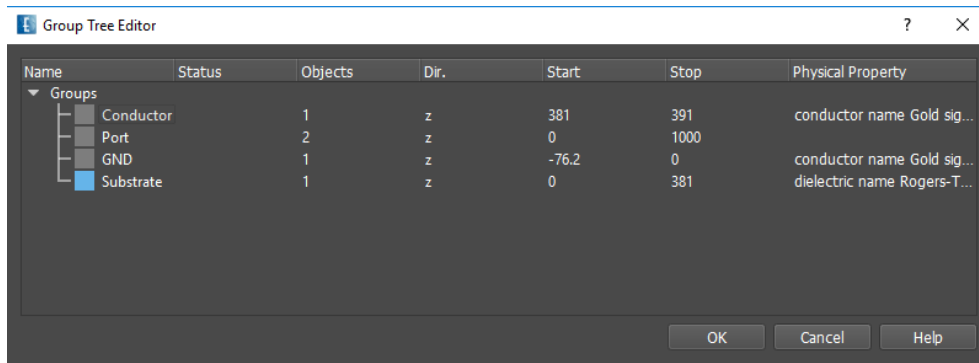


Figure 14.3.: Group Tree Editor

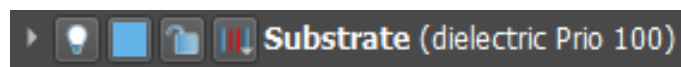


Figure 14.4.: Icons for single group

## 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**.

### 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

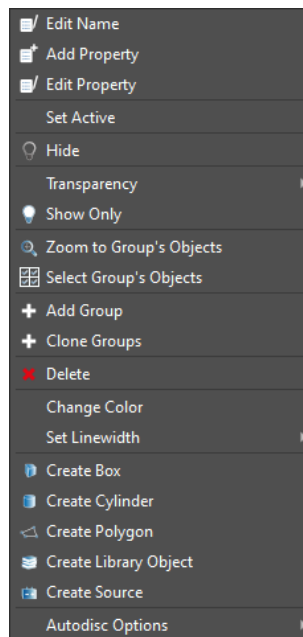


Figure 14.5.: Group menu

### 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.

# 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 - Components - 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

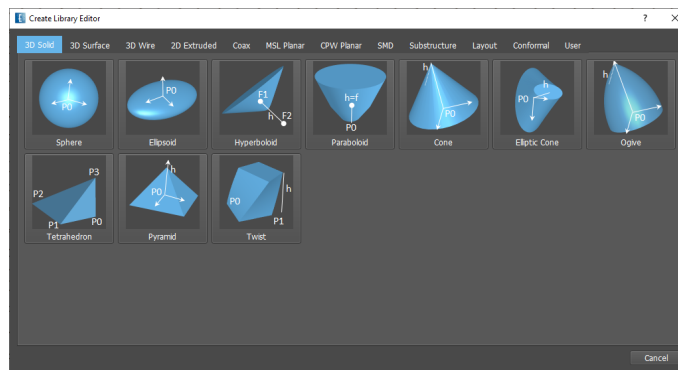











Figure 15.1.: View of 3D Solid Library

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.

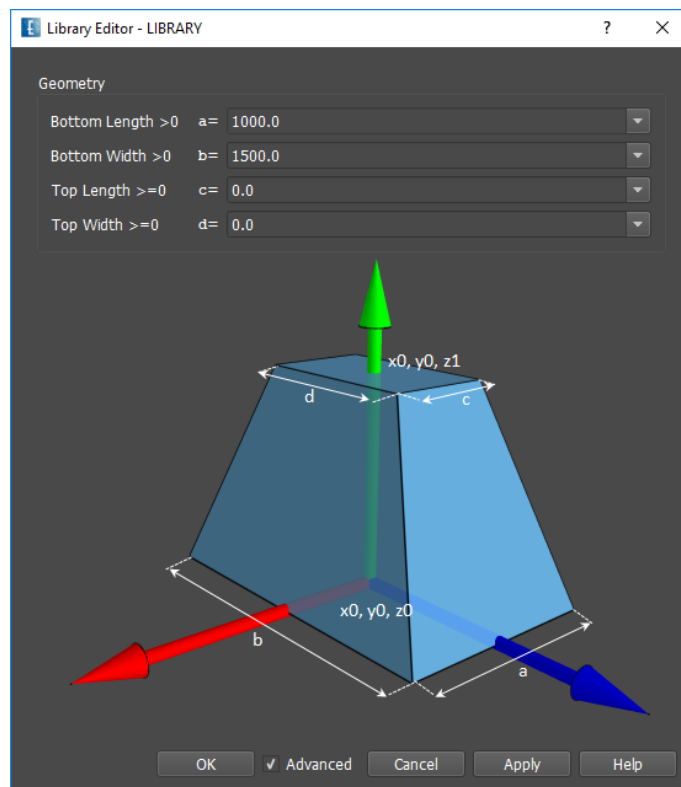


Figure 15.2.: Parameters of the pyramid

## 15.2.2. 3D Surface

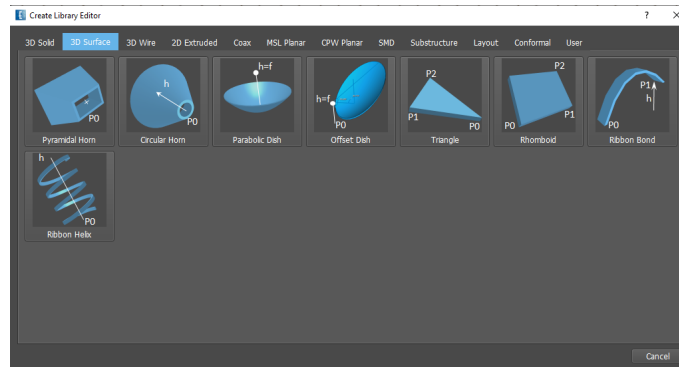


Figure 15.3.: View of 3D Surface Library

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.

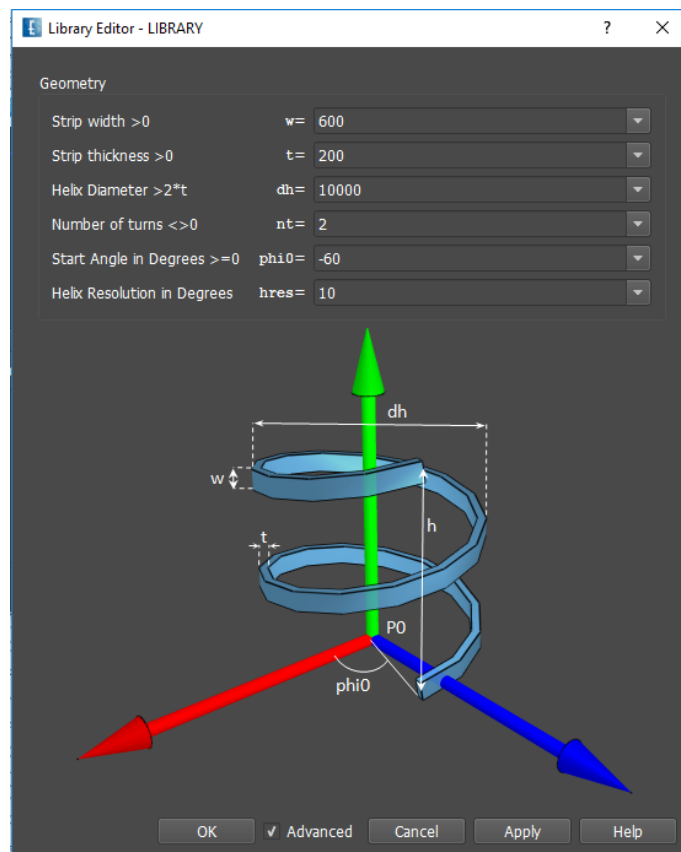


Figure 15.4.: Parameters of the ribbon helix

### 15.2.3. 3D Wire

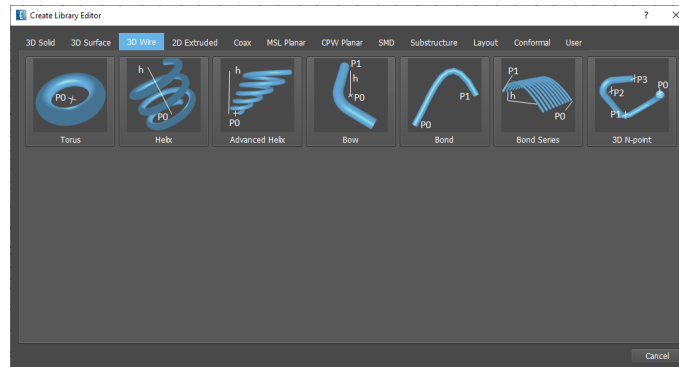


Figure 15.5.: View of 3D Wire Library

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  $\phi_{hs}$ , displacement shift and height  $h$ ) with normal vector perpendicular to  $h$ . Helix diameters at bottom and top ( $dh_0$  and  $dh_1$ ) may be different. Number of turns  $nt$  may be fractional, start angle  $\phi_{h0}$  and shift angle  $\phi_{hs}$  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

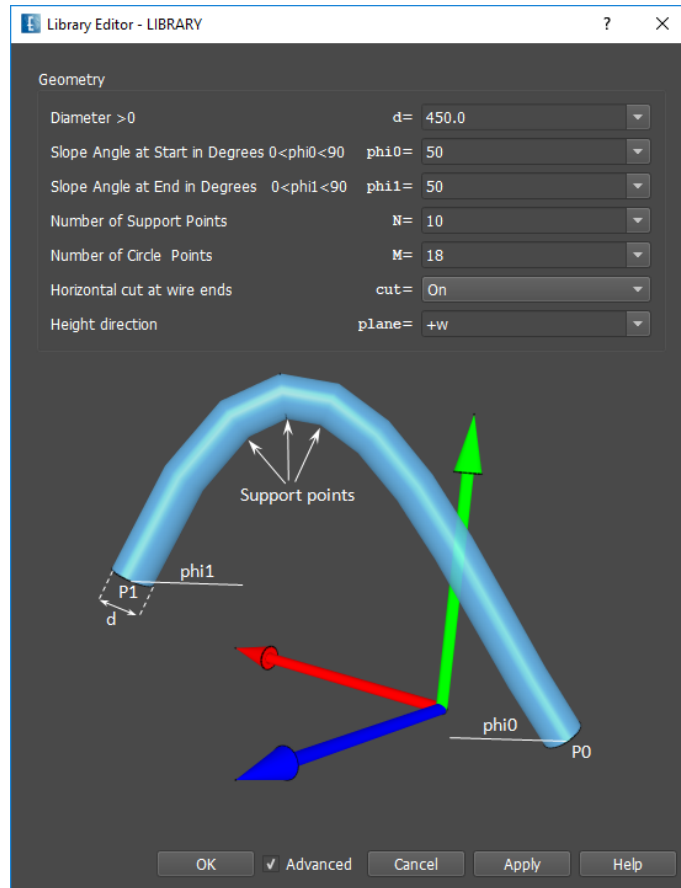


Figure 15.6.: Parameters of the bond wire

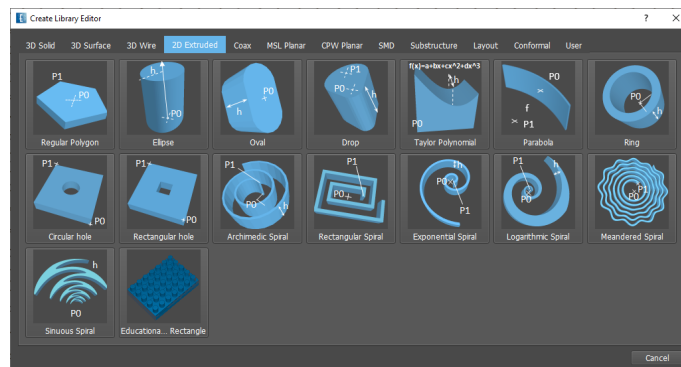


















Figure 15.7.: View of 2D Extruded Library

-  **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 consisting of inner and outer conductors and dielectric intermediate. 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.

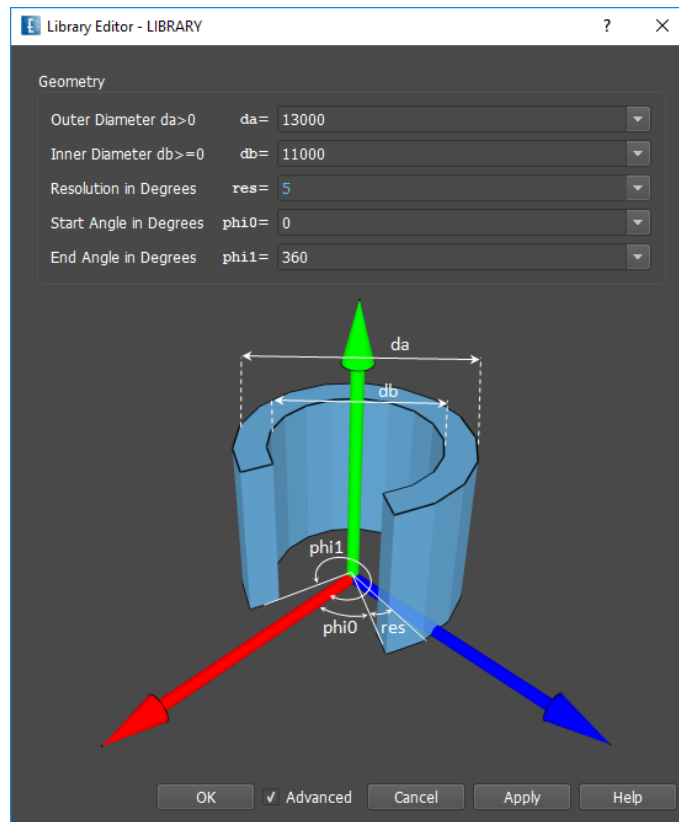


Figure 15.8.: Parameters of the ring

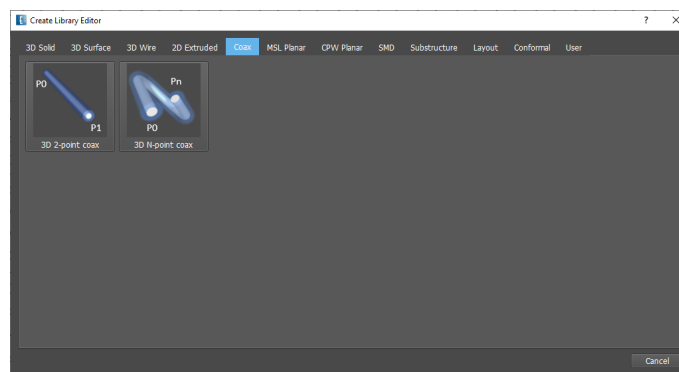


Figure 15.9.: View of Coaxial Library



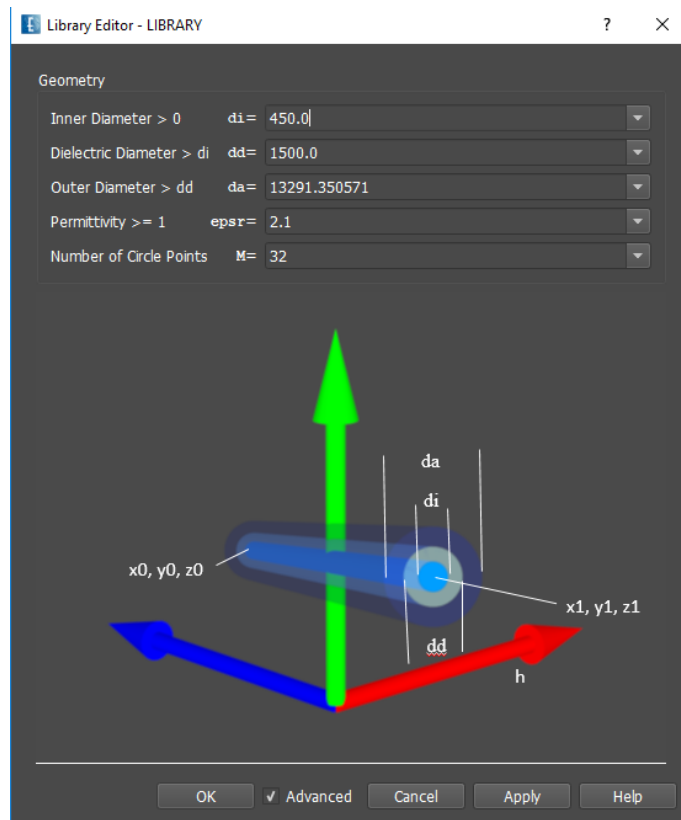


Figure 15.10.: Parameters of the 2-point coaxial element

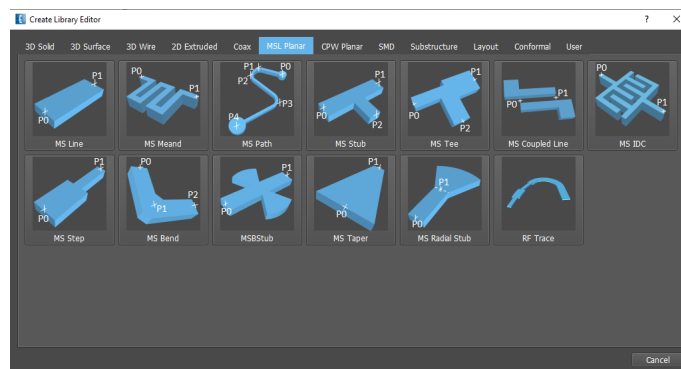















Figure 15.11.: View of MSL Planar Library




### 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.
-  **MSSStub** 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.

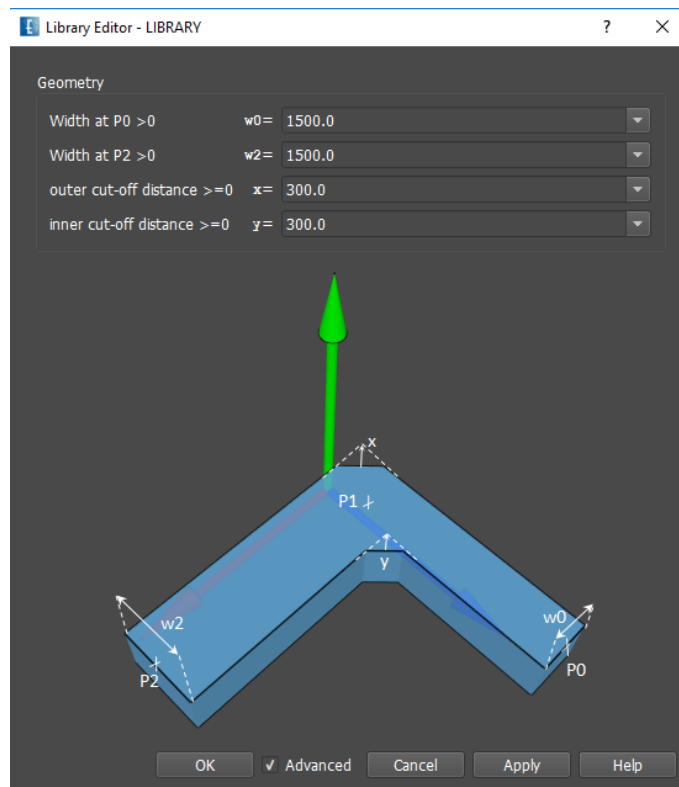


Figure 15.12.: Parameters of the MSL Bend

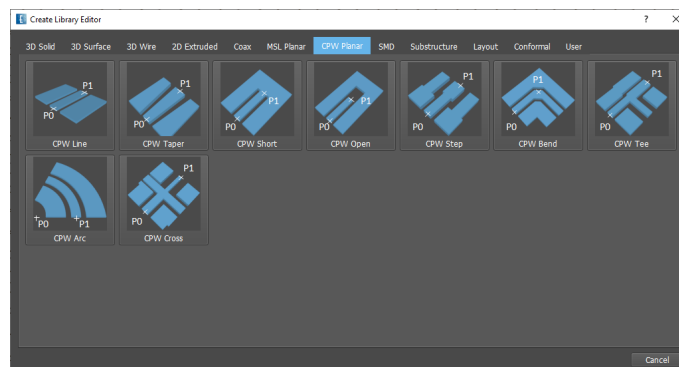


Figure 15.13.: View of CPW Planar Library

- **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.

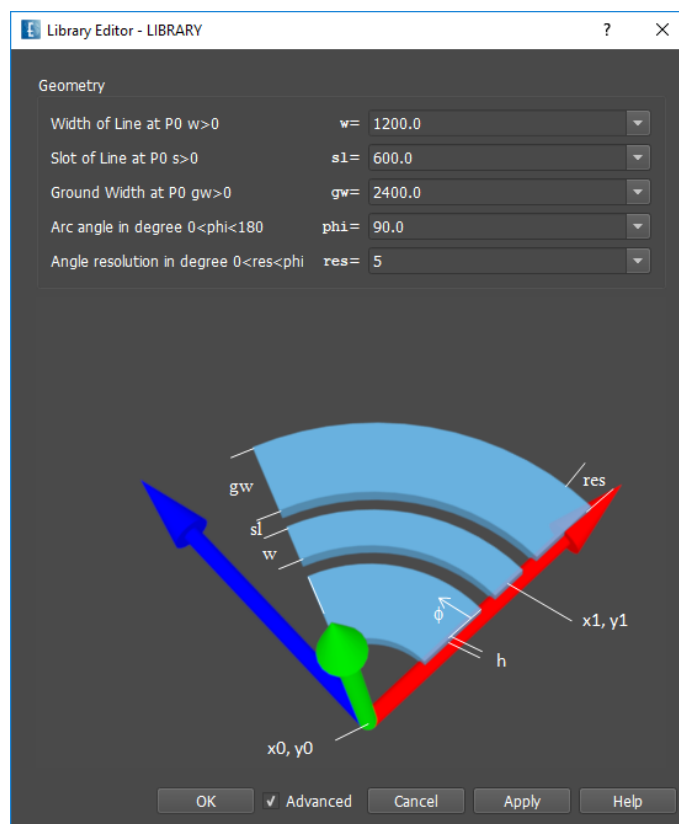


Figure 15.14.: Parameters of the circular CPW arc

### 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 properties. For compatibility reasons legacy SMD objects are still available.

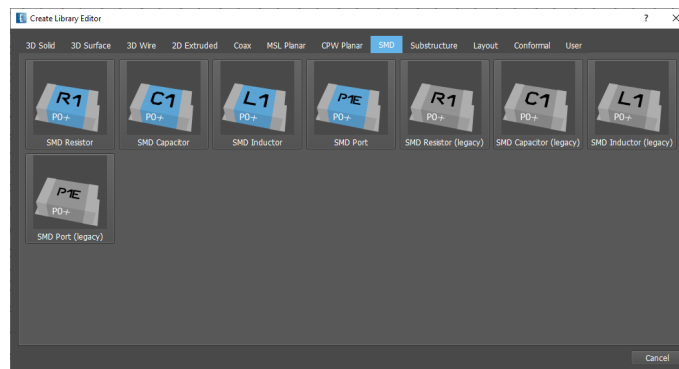



Figure 15.15.: View of SMD Library

- **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 capacitance can be set in Parameters.
- **SMD Inductor** The point defines the midpoint of the SMD. The inductance can 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.

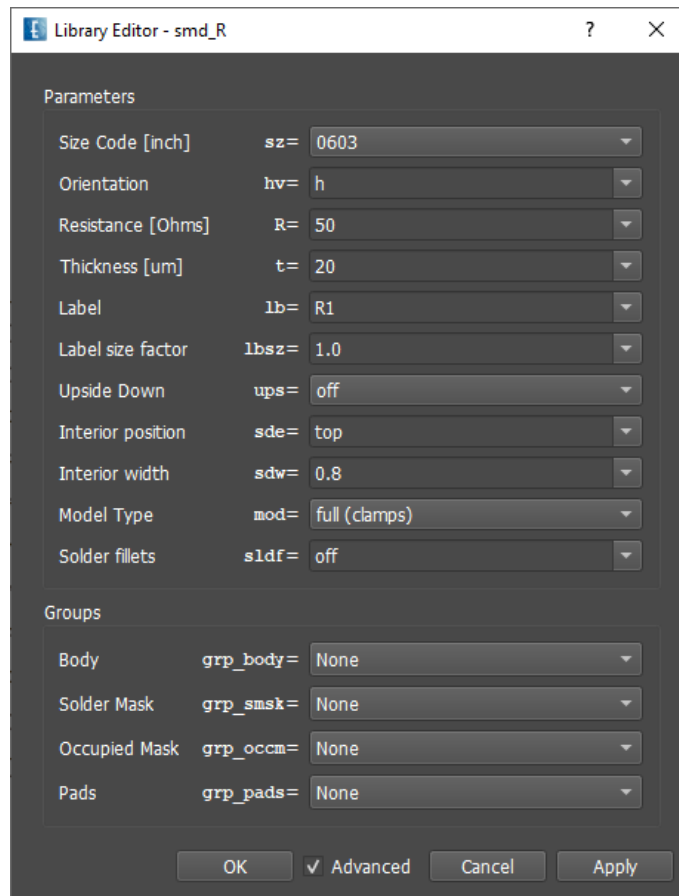


Figure 15.16.: Parameters of the SMD element

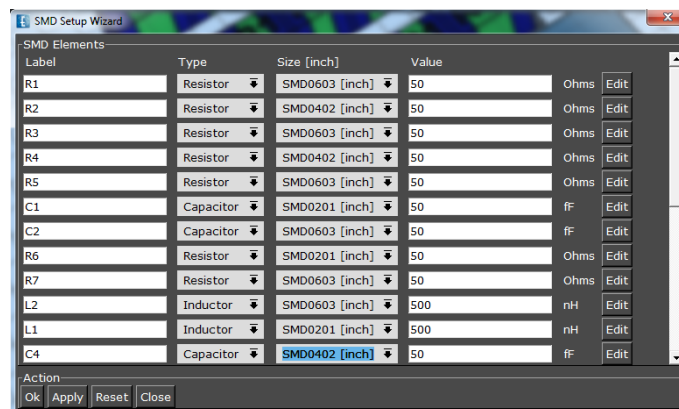


Figure 15.17.: SMD Setup Wizard

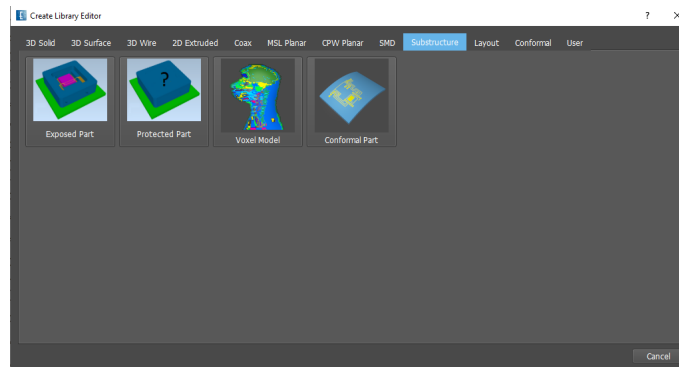


Figure 15.18.: View of Substructure Library

### 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**

## 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



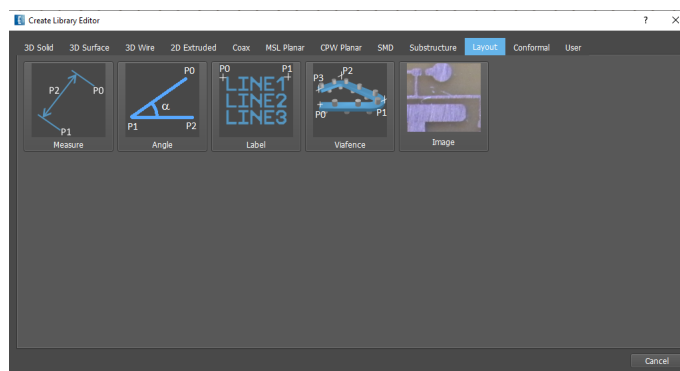







Figure 15.19.: View of Layout Library

### 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.
-  **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.

### 15.2.11. Conformal

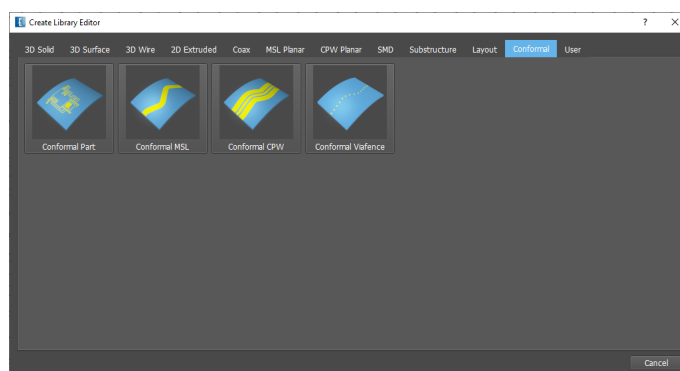


Figure 15.20.: View of Conformal Library

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 Viefence** Create a viefence 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.

### 15.2.12. User

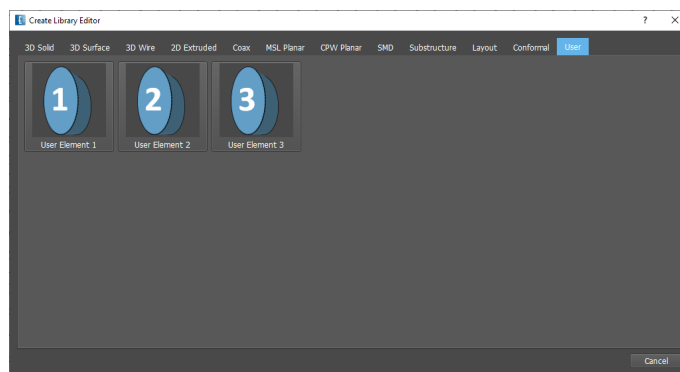


Figure 15.21.: User Defined Object Library

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
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
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.

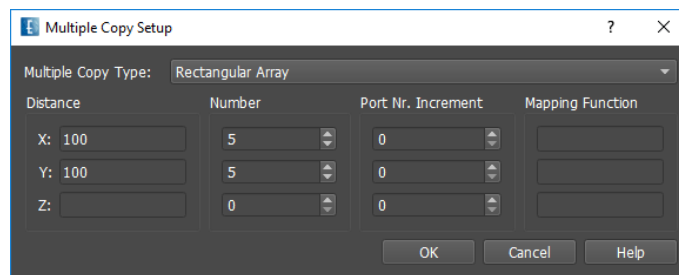


Figure 16.1.: Array copy wizard

#### 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  $x=30^\circ$  and 12 copies (Number=13). Note that the selected object is maintained as in all copy operations.

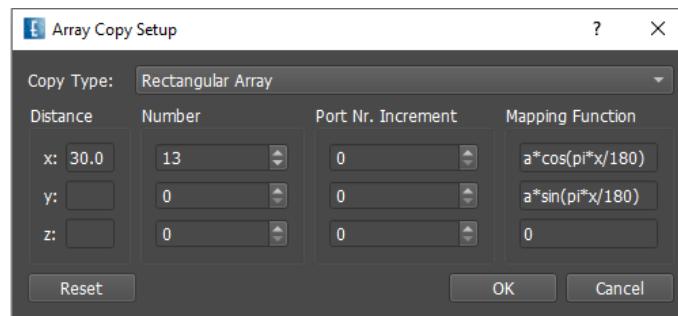



Figure 16.2.: Mapping example for array copy


### 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. The 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 (perpendicular) 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 single 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 (target-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.

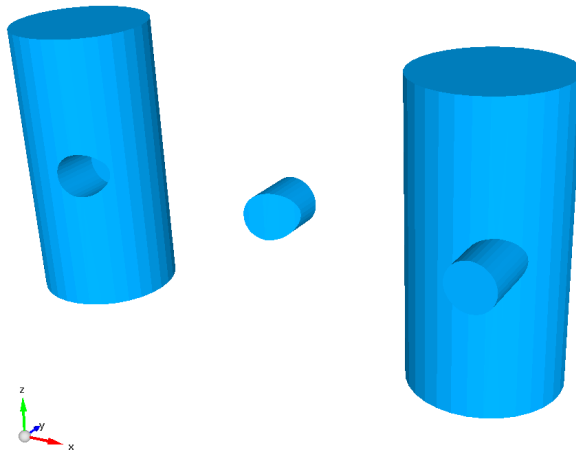




Figure 16.3.: Examples of Boolean operations Subtract, Intersect and Merge

# 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

#### Empire

- **Empire Meta File** Import a meta file (ending .gmf), 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.

#### 17.1.2. Empire


- **from Clipboard** Import selections from another project which are copied to Clipboard by **Ctrl c**
- **from File** Import project file, ending .gym
- **2D Points from File**

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 circuit 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

**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.

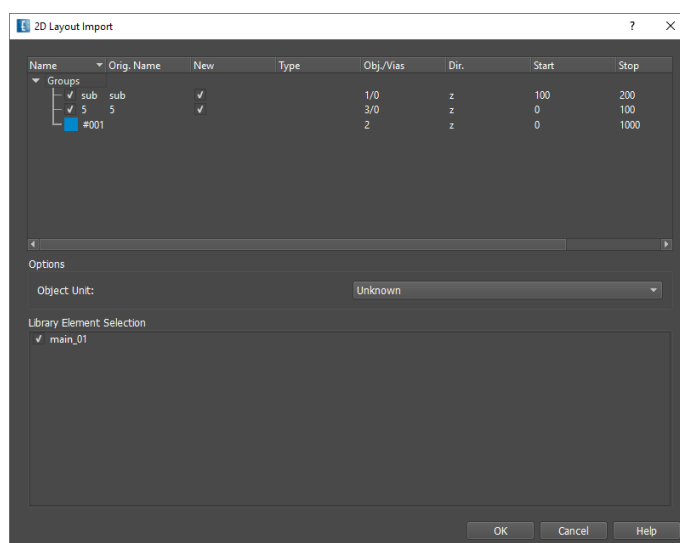


Figure 17.1.: Import wizard for DXF

**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

**17.1.5. 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 Tessellation 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

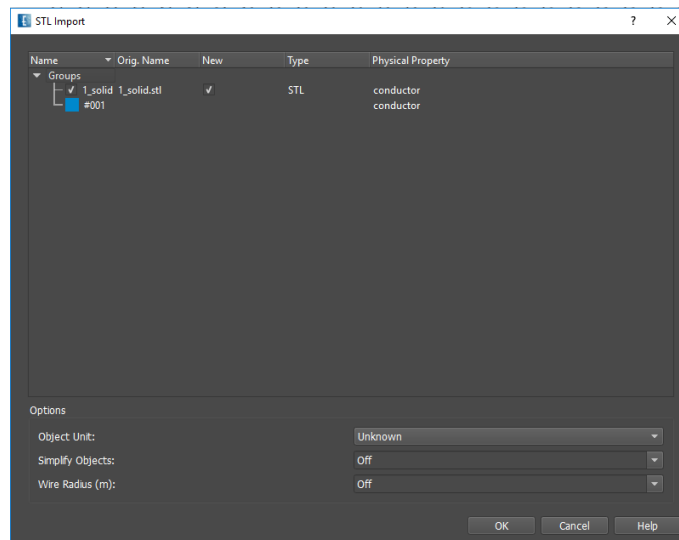



Figure 17.2.: Import wizard for STL

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

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)



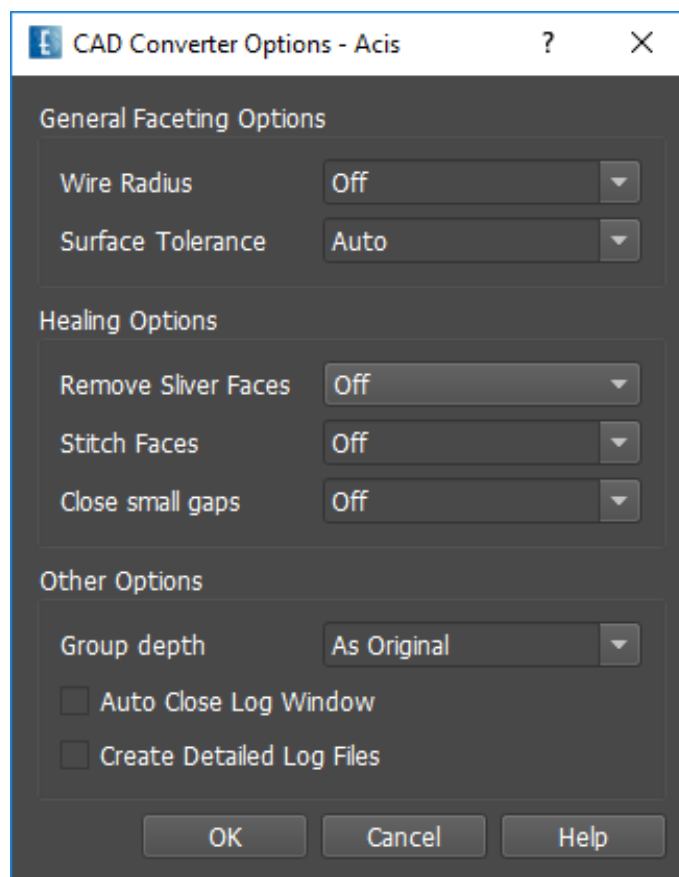



Figure 17.3.: CAD Converter Options for ACIS, CATIA, NX, etc.

- **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 stitched 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

### 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>.cst
- <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 details 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.

### 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.

### 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)
- **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

### 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**<sup>1</sup> 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.

<sup>1</sup>Each facets described as one planar BREP face

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

#### 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 applies.

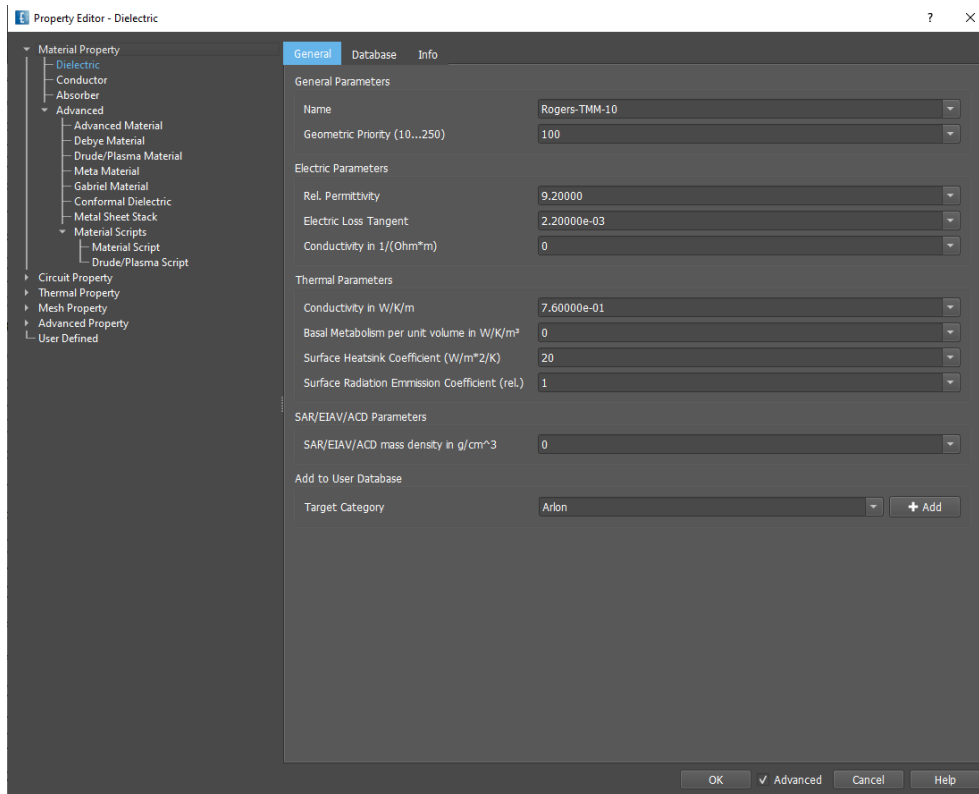


Figure 18.1.: Dielectric property setup

#### 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** -  $\epsilon_r \geq 1$
  - **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 one of these materials, open one of the entries by pressing ► and select one of the materials. Switch to **General** to verify the selection or to **Info** to see the frequency dependence.

### 18.2.2. Conductor

#### 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
  - **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  $\mu m$ , can be determined automatically. <sup>1</sup>
  - **Surface roughness** -  $0 \leq \rho < thickness$ , average roughness in  $\mu m$  using the Hammerstad & Jensen model:  $R_{\square}^r = R_{\square} \left( 1 + \frac{2}{\pi} \arctan(1.4 \frac{\rho^2}{a^2}) \right)$  where  $a$  is the skin depth and  $R_{\square}$  is the square resistance at the target frequency.
- Thermal Parameters

---

<sup>1</sup> where  $a$  is the skin depth  $a = \frac{1}{\sqrt{\pi f \mu \sigma}}$

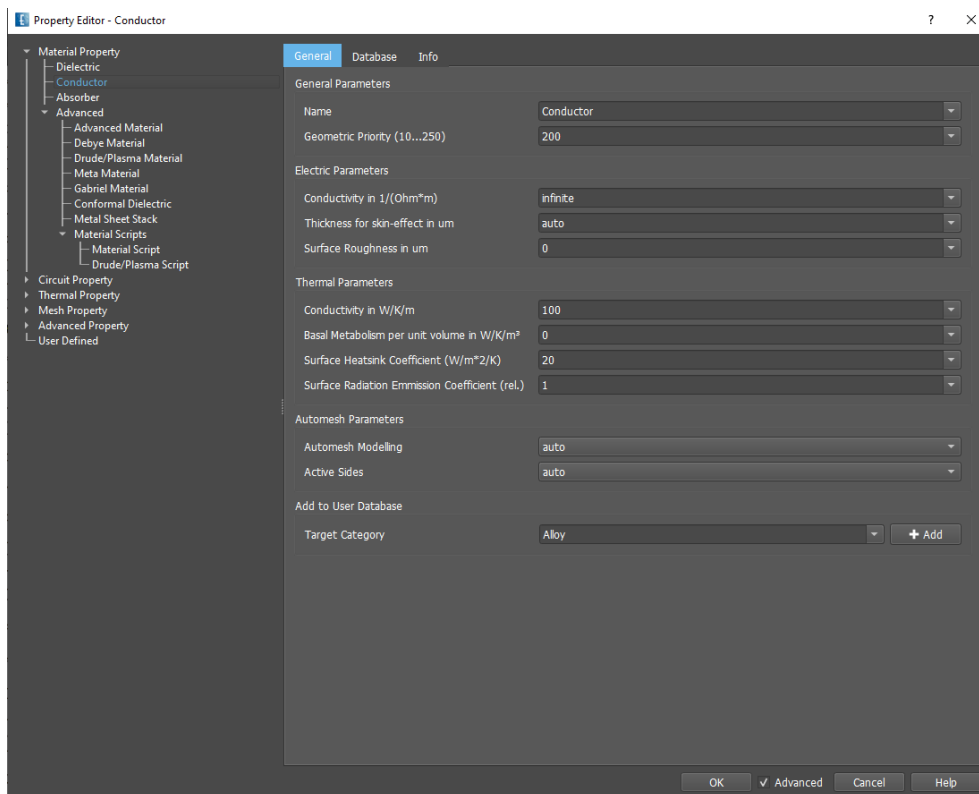


Figure 18.2.: Conductor property setup

- **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.

### Hint: Conductors

By default, conductors are considered as flat objects. To prevent this conductors can be assigned to be **bulk** objects or the objects should be assigned to **Dielectrics** with certain conductivity, e.g. to model sea water.

#### Database

To use predefined conductors open one of the entries by pressing ► and select one of the materials. Switch to **General** to verify the selection or to **Info** to see the frequency dependence.

### 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)

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

- 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
  - **Permittivity to match**  $\varepsilon_1 \geq 1$  Surrounding material of the absorber, e.g. air or substrate
  - **Permittivity of the Absorber**  $\varepsilon_2 \geq 1$ , e.g. ferrite permittivity
  - **Absorption abs-**  $\geq 0$ , 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.

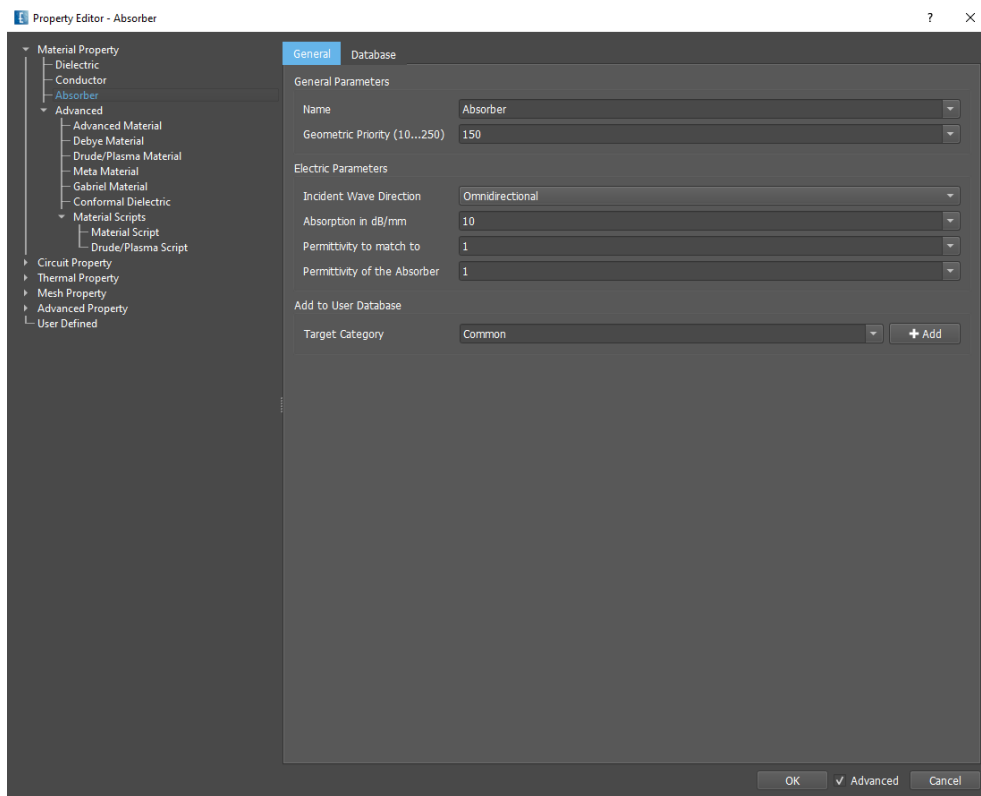


Figure 18.3.: Absorber property setup

## 18.2.4. Advanced - Material Object

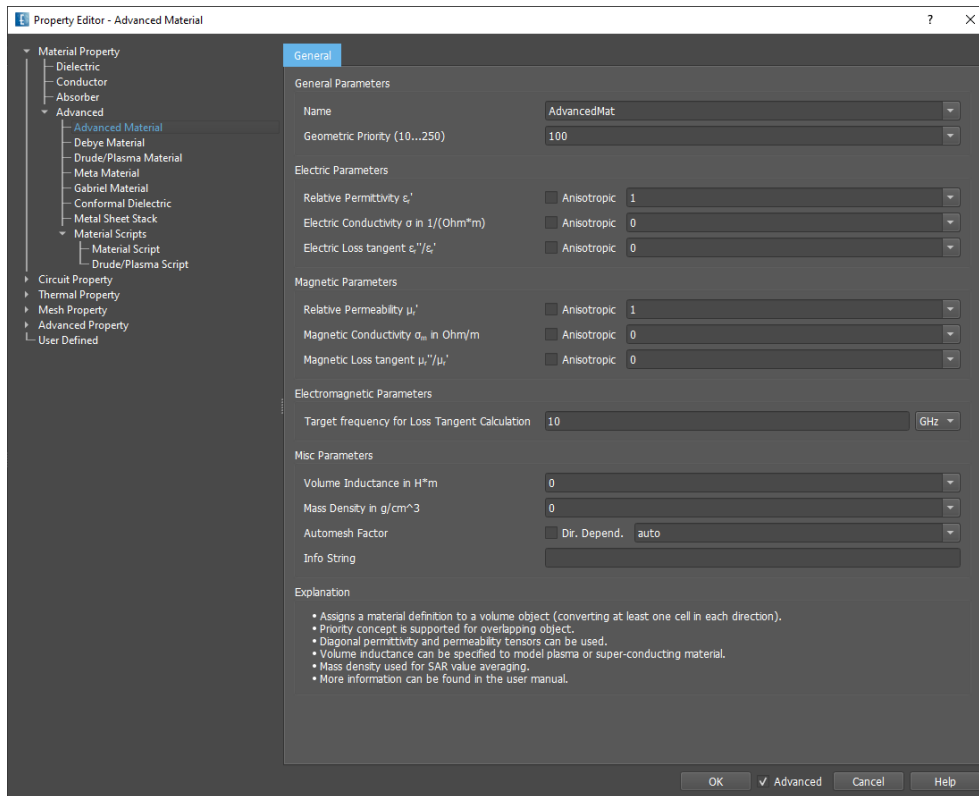


Figure 18.4.: Advanced Material property setup

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

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

- General Parameters
  - **Name** - A unique name for display in Group
  - **Geometric Priority** -  $10 < Integer < 250$ , used for intersecting objects
- Electric Parameters
  - **Rel. Permittivity** -  $\epsilon_r \geq 1$
  - **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  $\ell$  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  $L_v = -\frac{1}{\epsilon_0(\epsilon_{Plasma}+1)\omega^2}$
  - **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

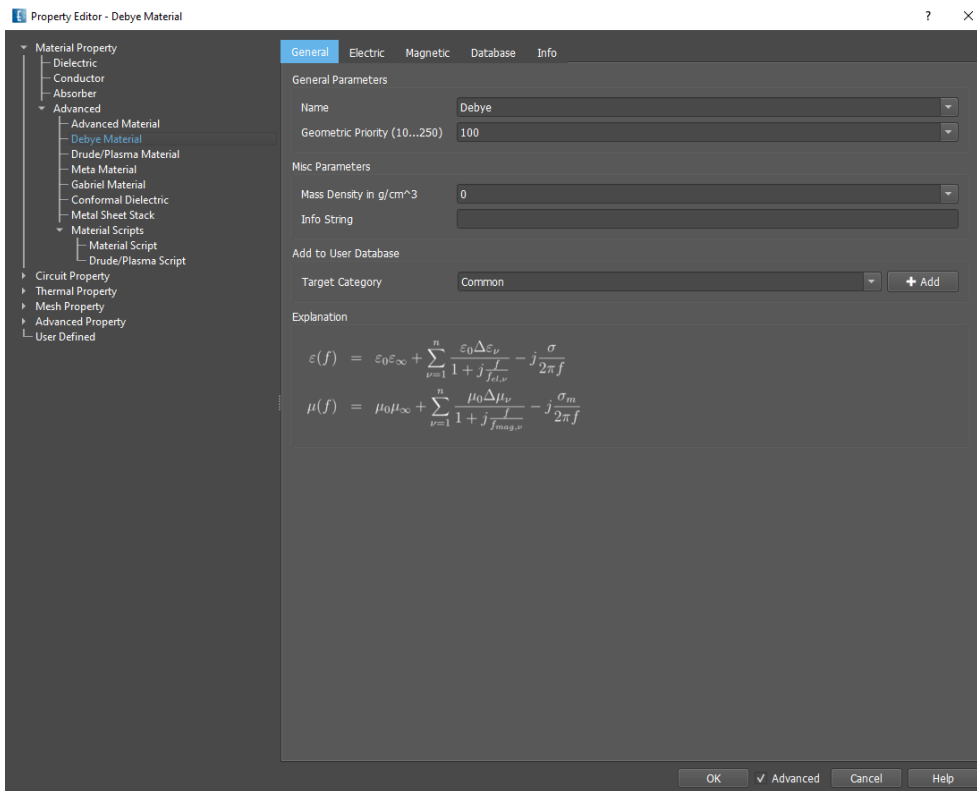


Figure 18.5.: Advanced Debye Material property setup

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

$$\epsilon(f) = \epsilon_0 \epsilon_\infty + \sum_{\nu=1}^n \frac{\epsilon_0 \Delta \epsilon_\nu}{1 + j \frac{f}{f_{el,\nu}}} - j \frac{\sigma}{2\pi f} \quad \mu(f) = \mu_0 \mu_\infty + \sum_{\nu=1}^n \frac{\mu_0 \Delta \mu_\nu}{1 + j \frac{f}{f_{mag,\nu}}} - j \frac{\sigma_m}{2\pi f}$$

- $\epsilon_\infty$  is the permittivity for  $f \rightarrow \infty$
- $f_{el,\nu}$  are the electric pole frequencies

- $\Delta\varepsilon_\nu$  are the relative changes of permittivity
- $\sigma$  is an additional electric conductivity
- $\mu_\infty$  is the permeability for  $f \rightarrow \infty$
- $f_{mag,\nu}$  are the magnetic pole frequencies
- $\Delta\mu_\nu$  are the relative changes of permeability
- $\sigma_m$  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 dependence of the created model.

- General Parameters
  - **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, conductivity, 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

### 18.2.6. Advanced - Drude Plasma

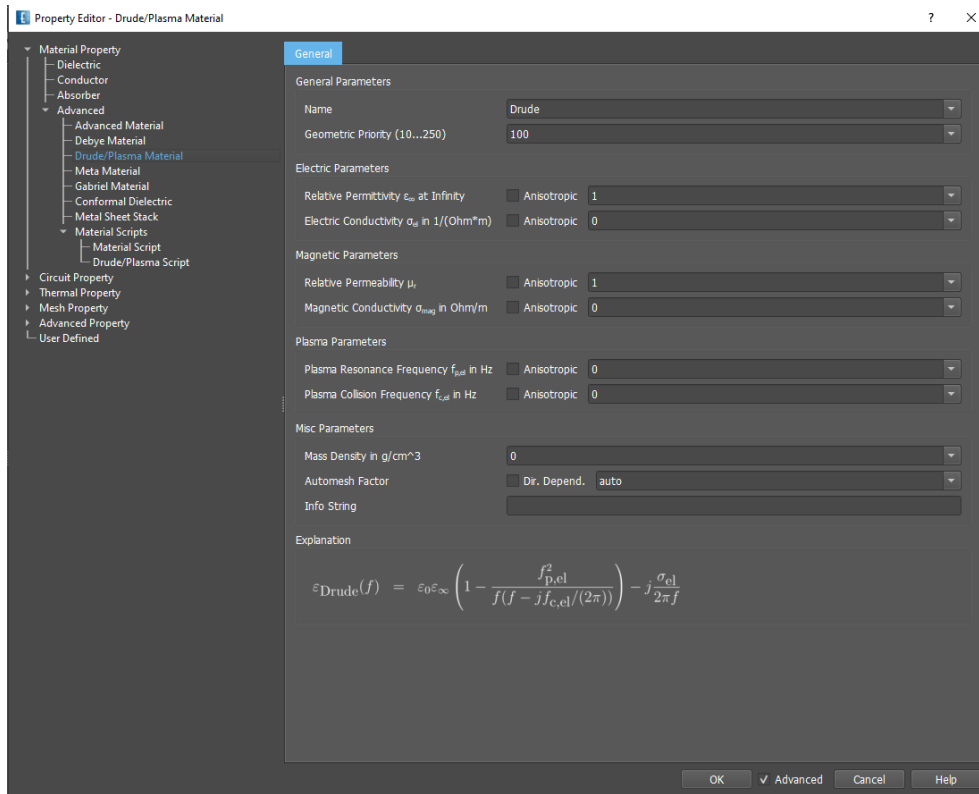


Figure 18.6.: Advanced Drude Material property setup

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:

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

Where the target frequency  $f$  is used in the simulation and  $\varepsilon$  and  $\sigma$  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

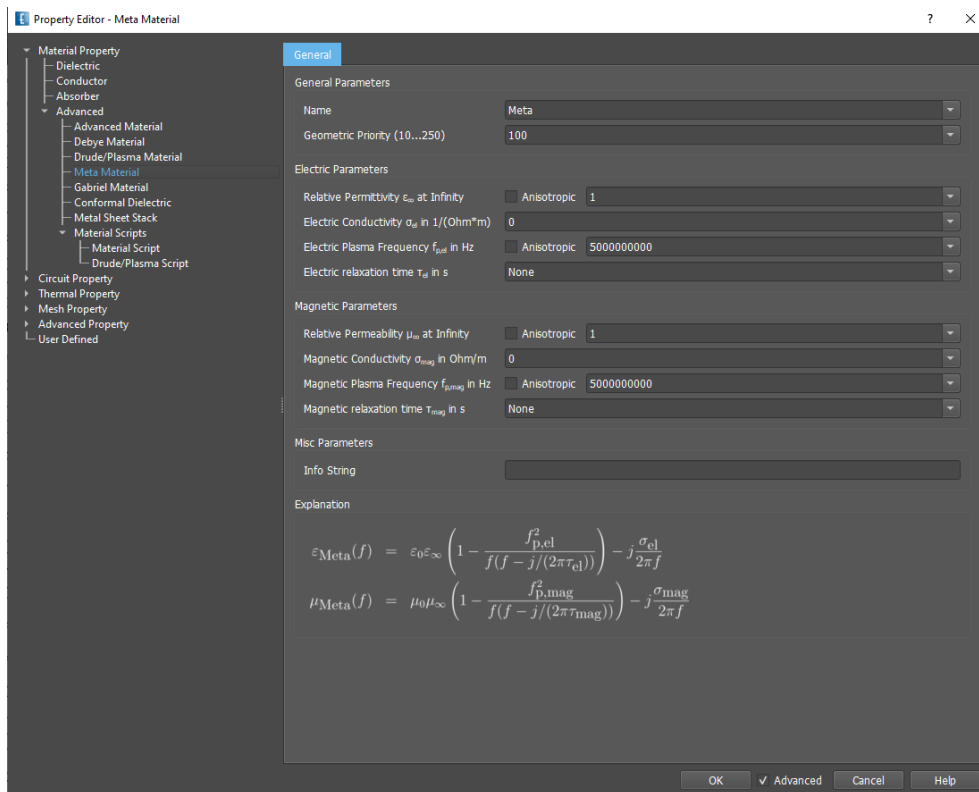


Figure 18.7.: Advanced Meta Material property setup

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  $\epsilon_{eff}$  as well as a negative permeability  $\mu_{eff}$  can be observed. At the transition frequency the effective material parameters

$\varepsilon_{eff}$  and  $\mu_{eff}$  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
  - **Geometric Priority** -  $10 < Integer < 250$ , used for intersecting objects
- Electric Parameters
  - **Rel. Permittivity** -  $\varepsilon_r$  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  $\tau_{el}$
- Magnetic Parameters
  - **Rel. Permeability** -  $\mu_r$  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  $\tau_{mag}$
- Misc Parameters
  - **Info String** for advanced usage

### 18.2.8. Advanced - Gabriel Material

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

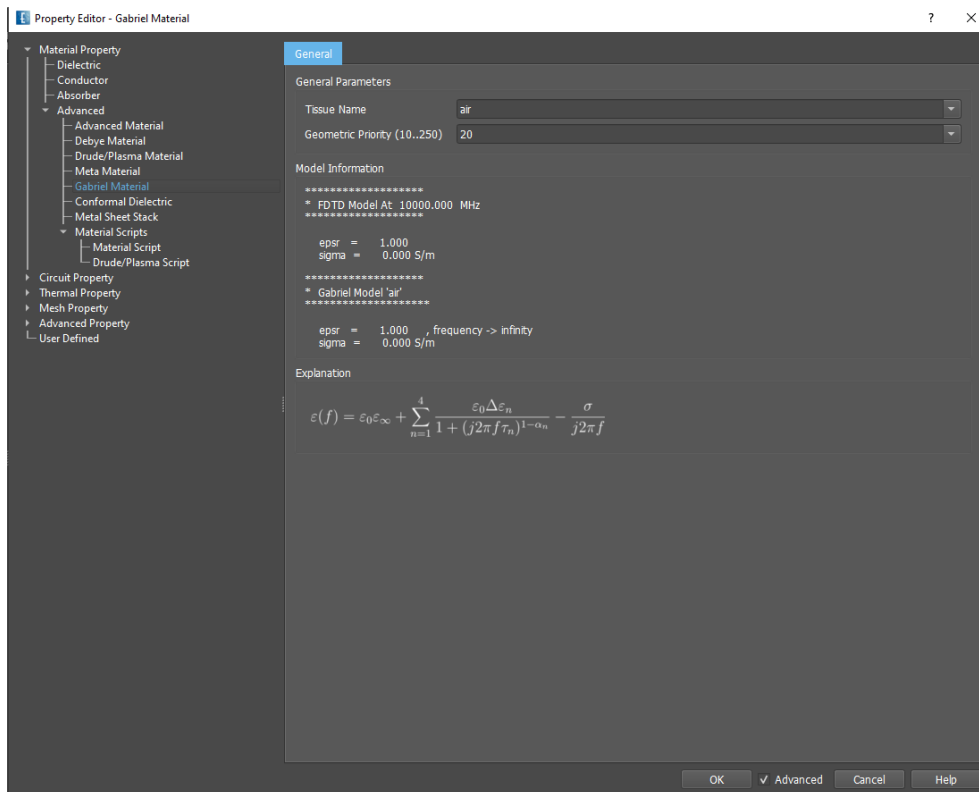


Figure 18.8.: Advanced Gabriel Material property setup

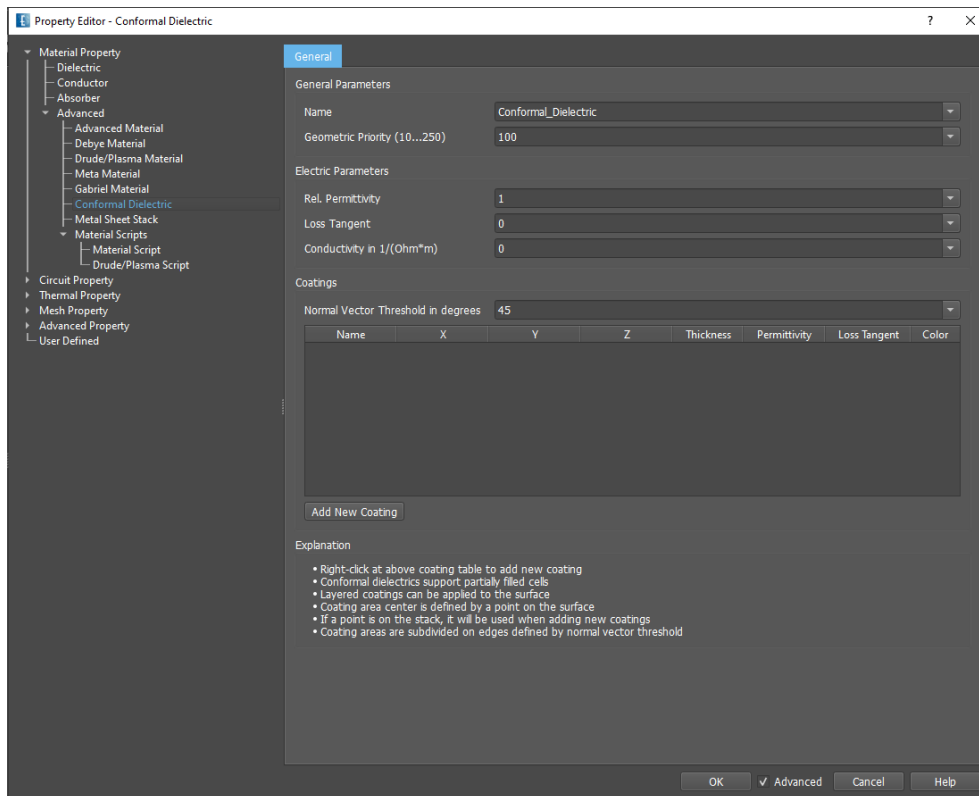


Figure 18.9.: Advanced Conformal Dielectric property setup

### 18.2.9. Advanced - Conformal Dielectrics

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** -  $\epsilon_r \geq 1$
  - **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** -  $\epsilon_r \geq 1$  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

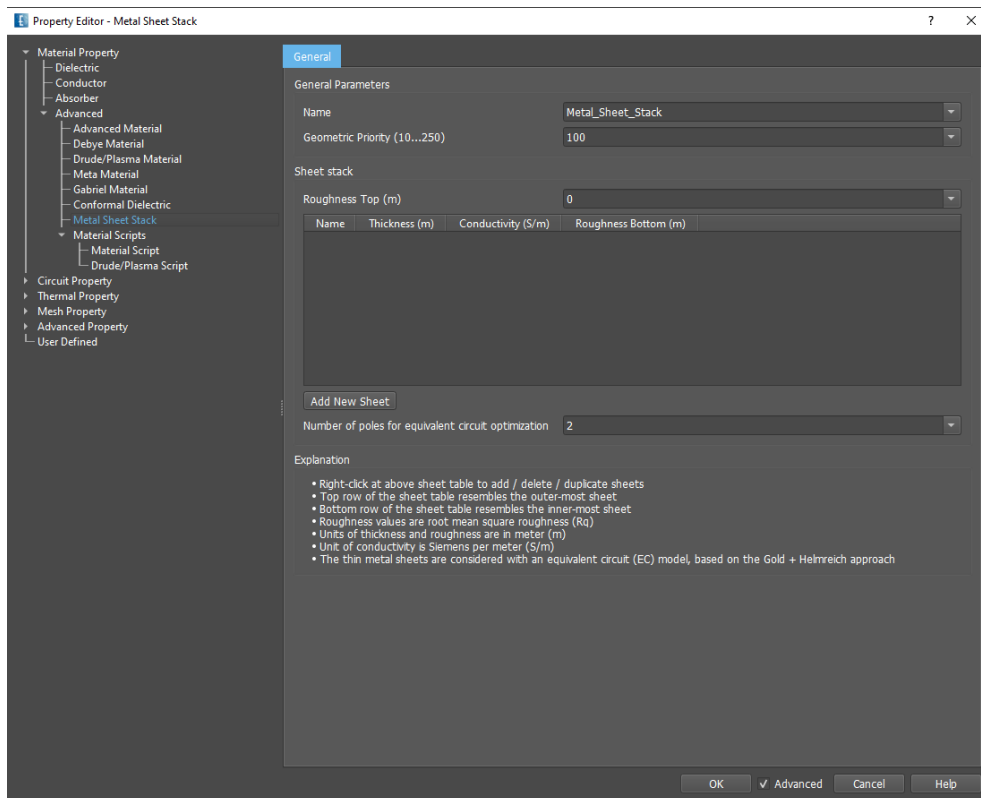


Figure 18.10.: Advanced Metal Sheet Stack property setup

### 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 included 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  $\Omega$ .
- **Capacitor** - Value ( $> 0$ ) has to be entered in Farad. <sup>2</sup>
- **Inductor** - Value ( $> 0$ ) has to be entered in Henry. <sup>3</sup>
- **LRC circuit** - Parallel circuit of **Rp**, **Cp**, and **Rs-Ls**. Values ( $> 0$ ) or **None** have to be entered in Henry, Ohm or Farad.

---

<sup>2</sup>Maximum value can be determined from  $N_{timesteps}\delta t > 3\tau = 3RC$

<sup>3</sup>Maximum value can be determined from  $N_{timesteps}\delta t > 3\tau = 3\frac{L}{R}$



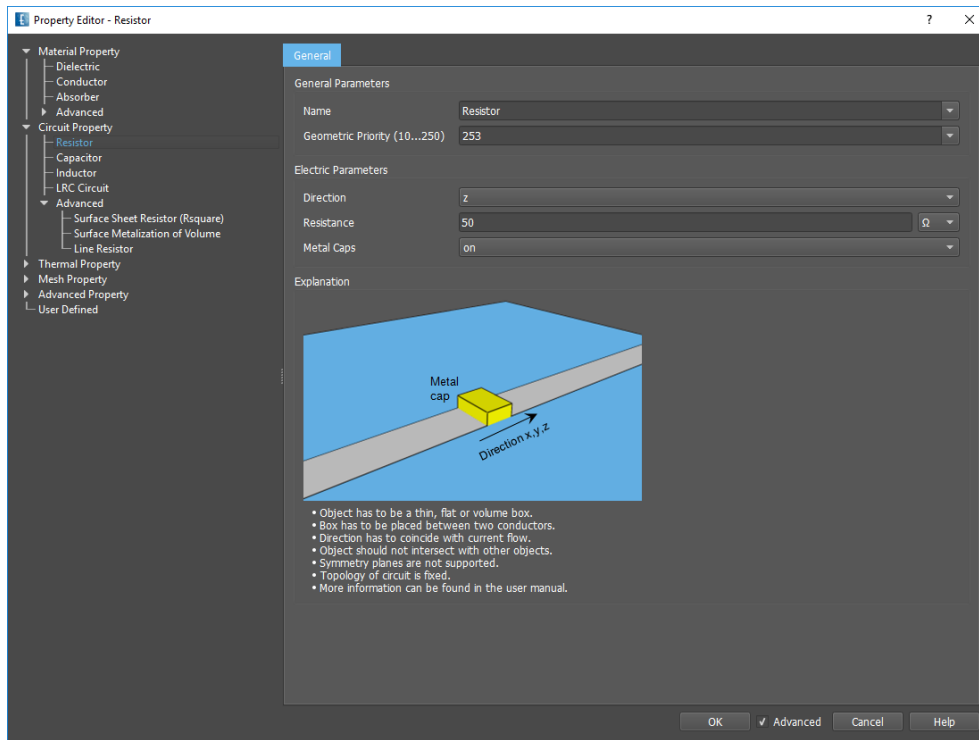


Figure 18.11.: Lumped resistor definition

### 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 resistor in  $\Omega$  can be defined
- The box must be thin, e.g. cover only one grid line in one direction.
- 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 metallization of volume objects (solids).

- **Metallization** - Surface will be metallized 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 constraint 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, e.g. use **Fine** instead of **Medium** if value is 2.
- **Max. Interval** Absolute refinement number, all grid lines in this area get a maximum spacing of the entered value.
- **Min. Interval** Absolute number, preventing cell spacing lower than this limit

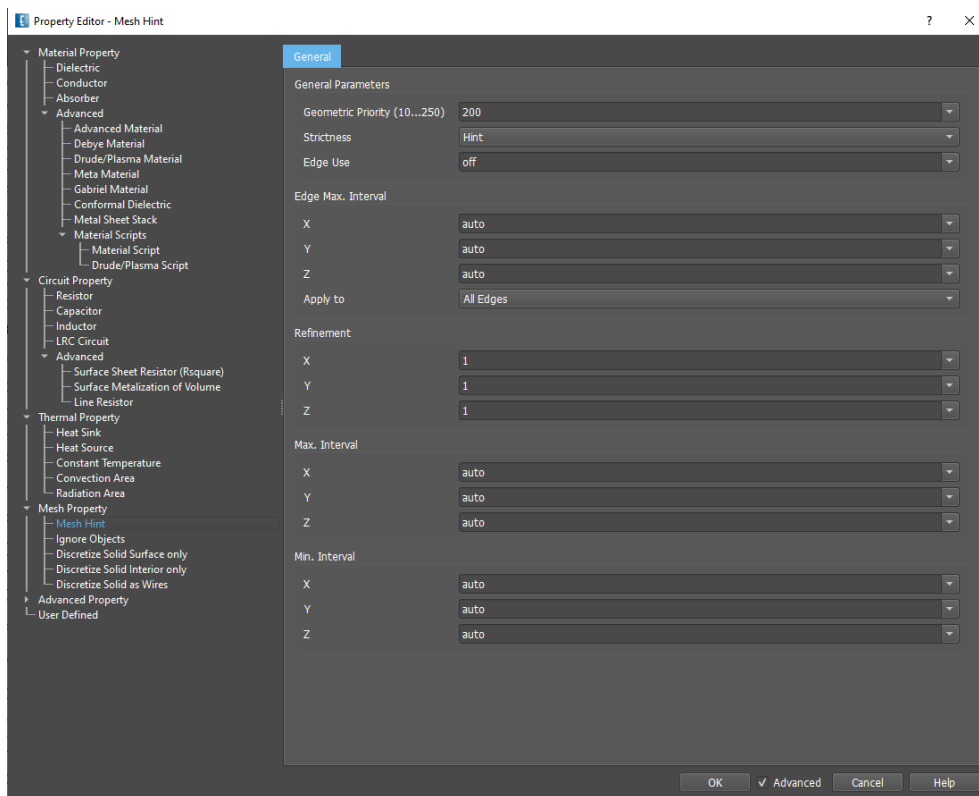


Figure 18.12.: Mesh hint definition

### 18.5.2. 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.

### 18.5.3. 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.

### 18.5.4. 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.

### 18.5.5. 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 ( $\epsilon_r$  and  $\sigma_e$ )
  - **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
  - **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**

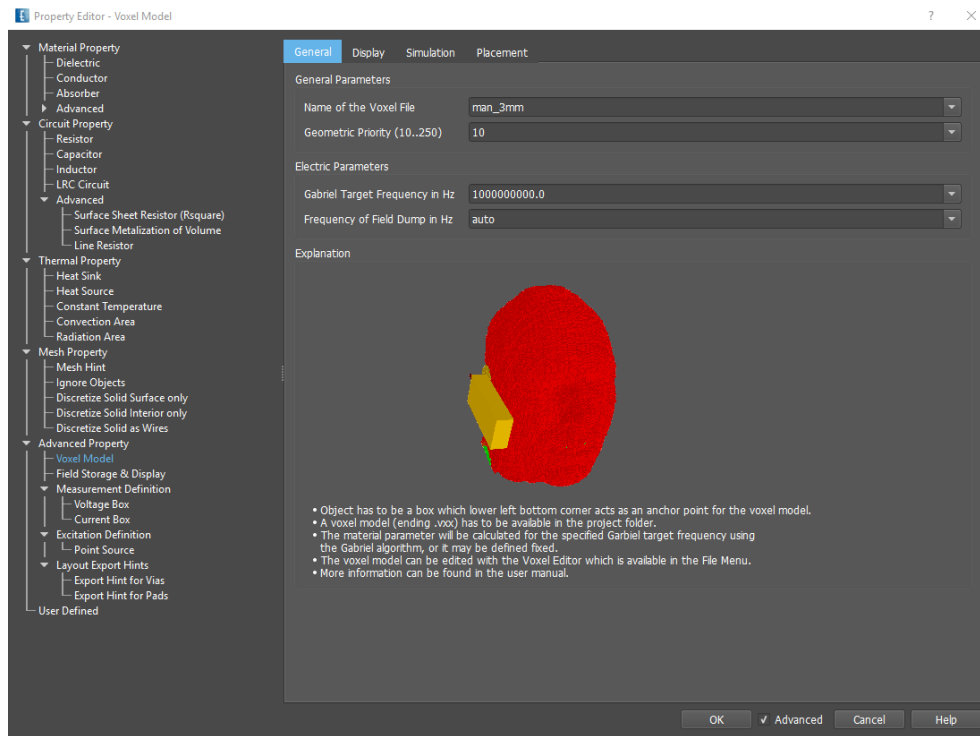


Figure 18.13.: Advanced property definition

- **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  $x_0, y_0, z_0$
  - **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. User defined

User defined **Text** property. Used for compatibility reasons or special purposes, e.g. to copy and paste properties from one group to another.

# 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: 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** .

## 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 .

## 19.1. EM Field

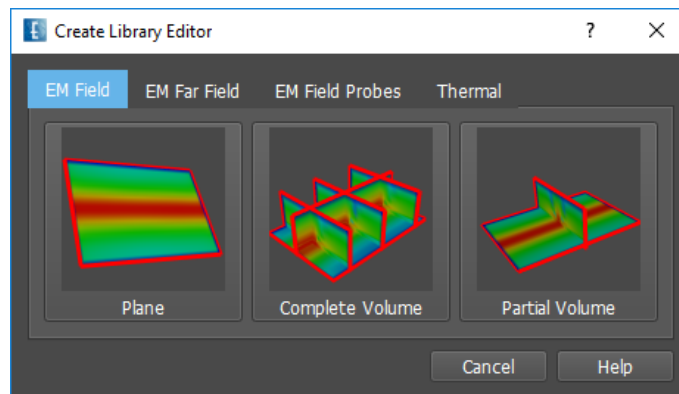


Figure 19.1.: Field Monitor EM Field

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

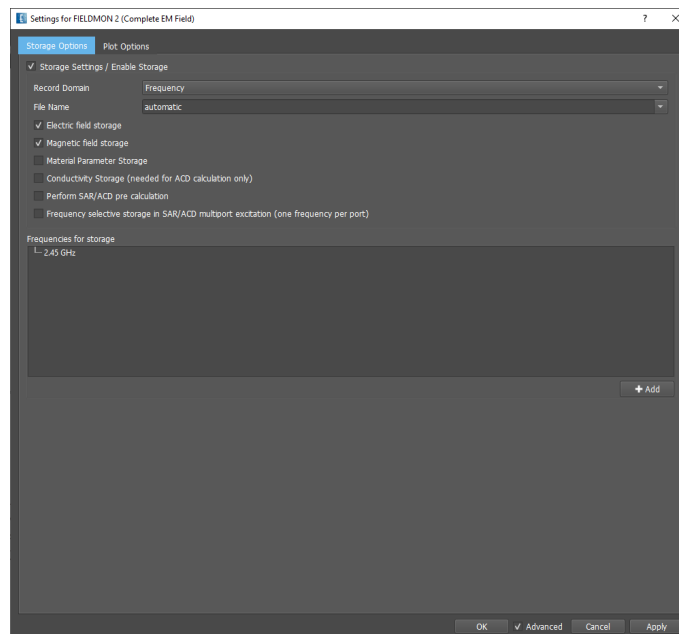


Figure 19.2.: Field dump storage definition

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.
- **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 pre calculation** A cache file will be created in postprocessing with the settings defined in 3D Plot options
- **Frequency selective storage in SAR/ACD multipoint 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)
- **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

- **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.  $NF \times Nx \times Ny \times Nz \times 48\text{Bytes}$  or  $NT \times Nx \times Ny \times Nz \times 24\text{Bytes}$  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.

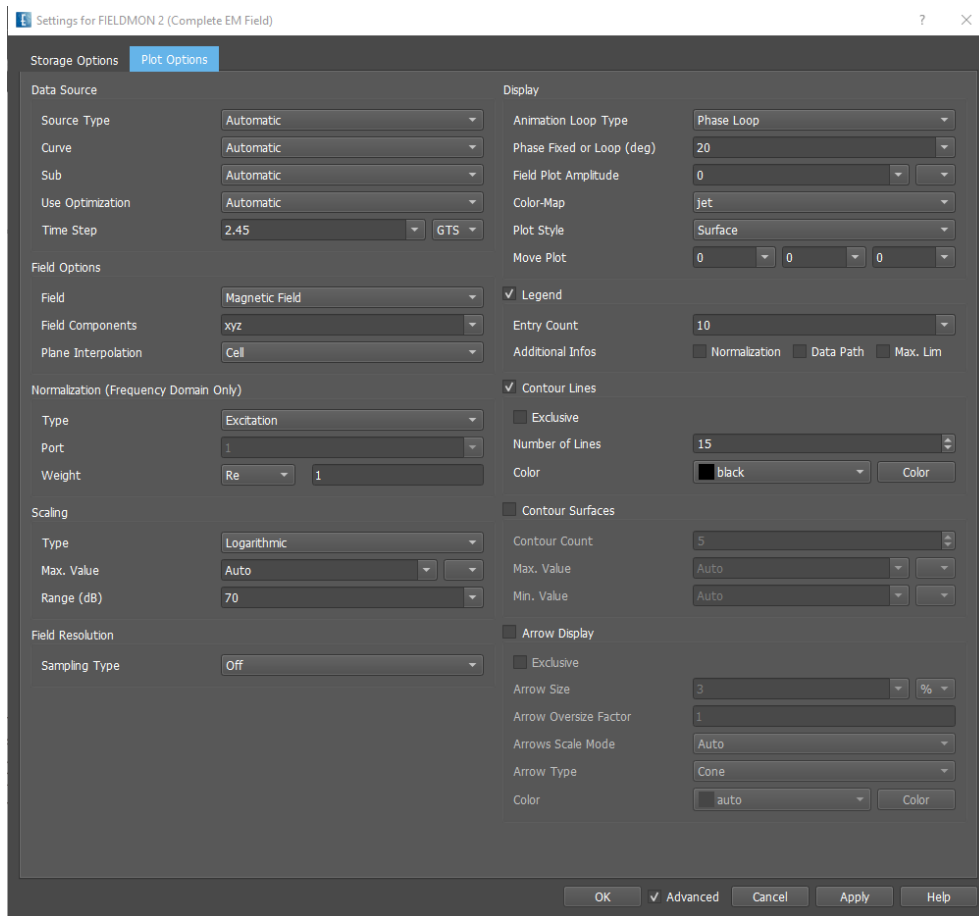
**19.1.2. 3D Plot Options**

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  $\epsilon_r$  - permittivity distribution if Material Storage is enabled
  - Conductivity  $\sigma$  - 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 maximum 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 Absorption 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 monitor
- **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 polygonal or STL facets
- **Export Animation** Save animation sequence as Stanford PLY, VTK polygonal 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

### 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



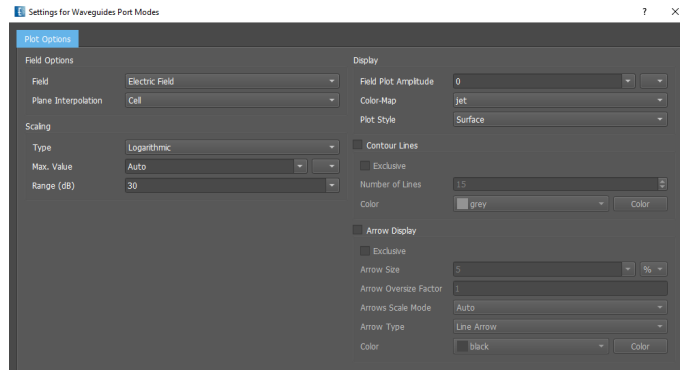


Figure 19.4.: Waveguide Port Mode Monitor

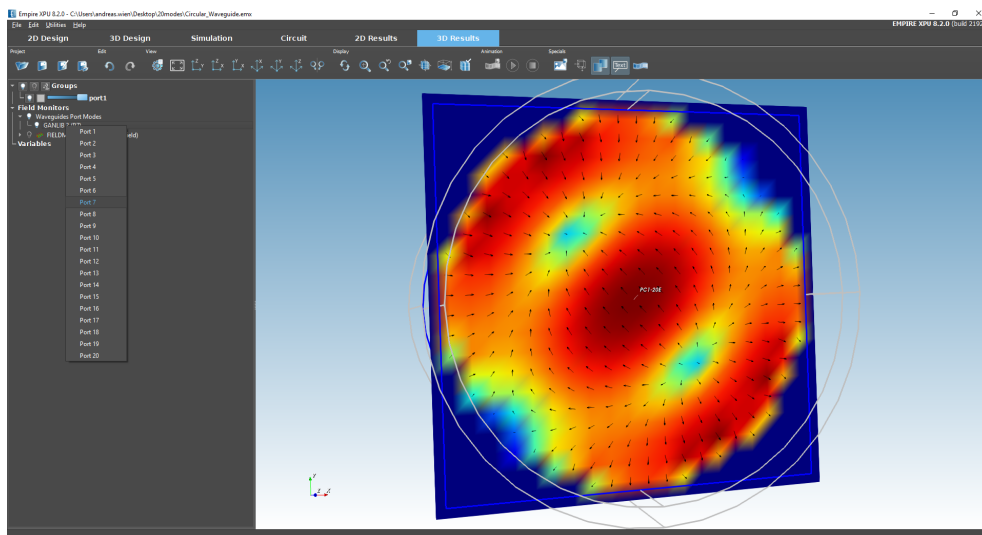


Figure 19.5.: Mode Display

### 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 maximum 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 quantities. 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.

## 19.3.1. Storage Options

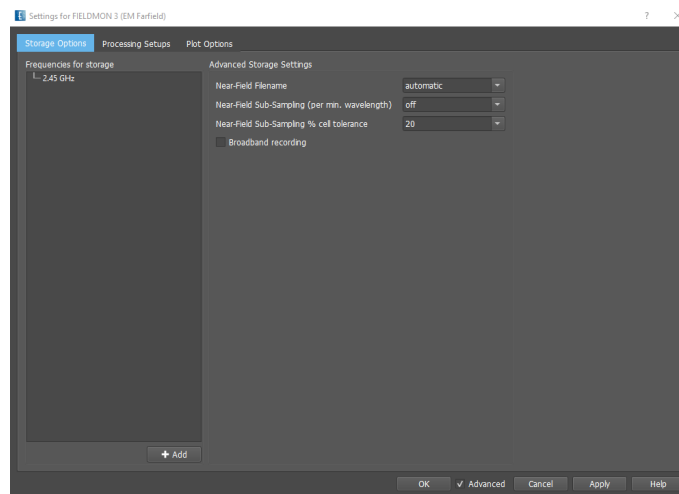


Figure 19.6.: Far-field storage definition

### 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 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

**Hint: Farfield box size**

The Farfield recording requires approx.  $NF \times (NxNy + NyNz + NzNx) \times 96\text{Bytes}$  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

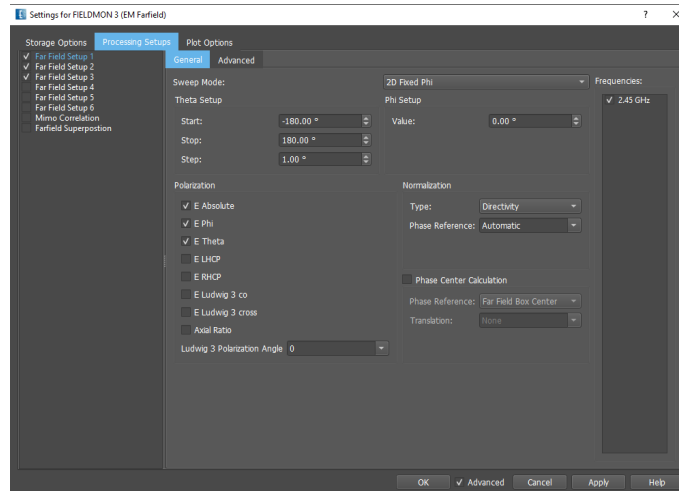


Figure 19.7.: Far field calculation

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  $\left| \frac{|eright|+|left|}{|eright|-|left|} \right|$
- **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 Directivity 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: Directivity 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 enabled 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  $E_{\phi}$ ,  $E_{\theta}$  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.

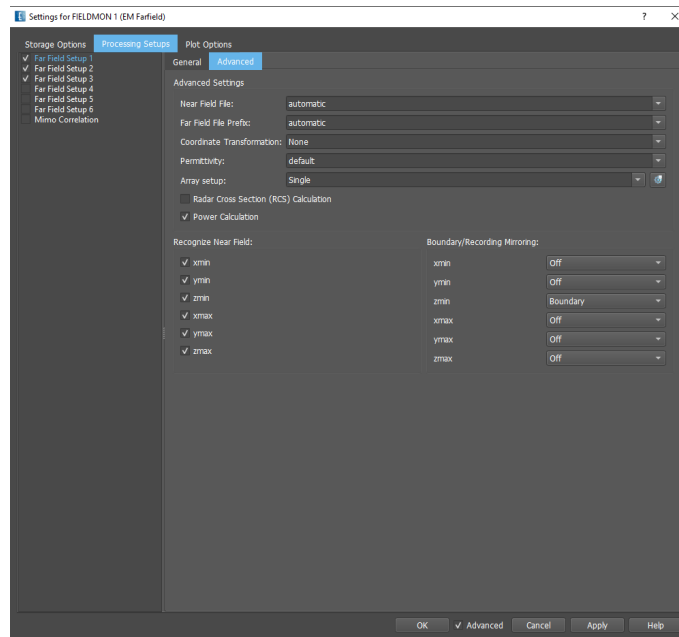


**Advanced**

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** -  $\zeta$  refers to  $\theta$  and angle  $\psi$  refers to the angle  $\phi$  in spherical coordinates
  - **Any Axis, any angle** - axis defined by x,y,z direction and rotation angle  $\alpha$ . (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 Hugins box, e.g. ground (adds 3 dB in far field calculation)
  - **Recording Box (Symmetry)** - Enable mirror for symmetry on a user-defined Hugins 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  $k - 1$  coefficients 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:

$$P_{\theta} = e^{-\left(\frac{\theta(\pi/2 - m_v)}{\sigma_v}\right)^2} \quad P_{\phi} = e^{-\left(\frac{\theta(\pi/2 - m_h)}{\sigma_h}\right)^2}$$

Here  $E_{\theta}, E_{\phi}$  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

$$\rho_e^{DCF} = \frac{|S_{i,i}^* S_{i,j} + S_{j,i}^* S_{j,j}|^2}{(1 - |S_{i,i}|^2 - |S_{j,i}|^2)(1 - |S_{i,j}|^2 - |S_{j,j}|^2)}$$



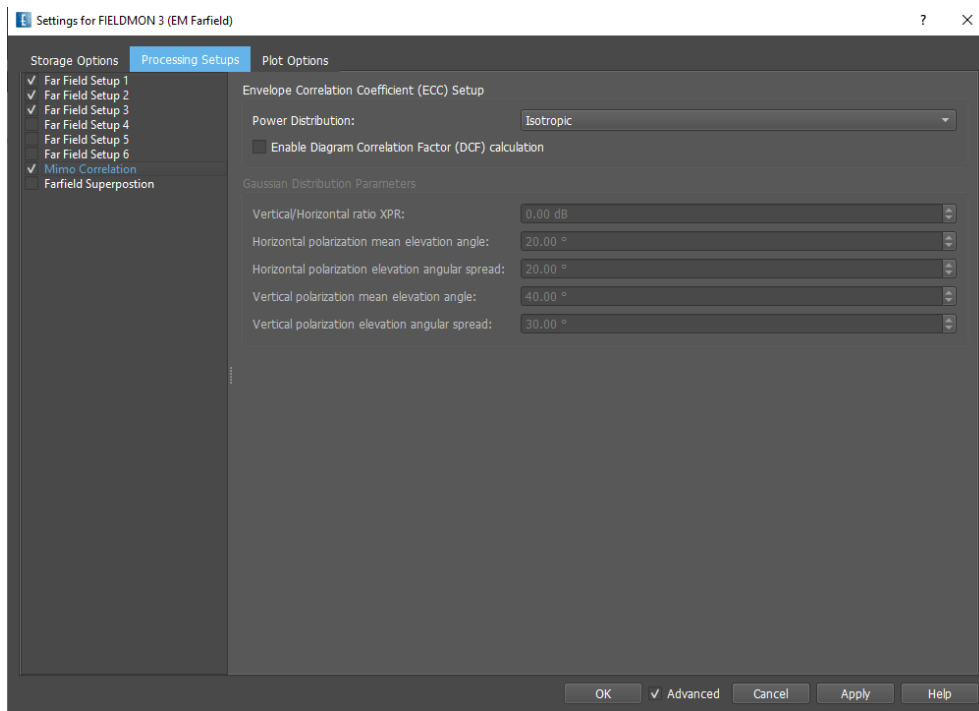


Figure 19.9.: Mimo Correlation Setup

### 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.  $1 + 0.5j$

### 19.3.5. 3D 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

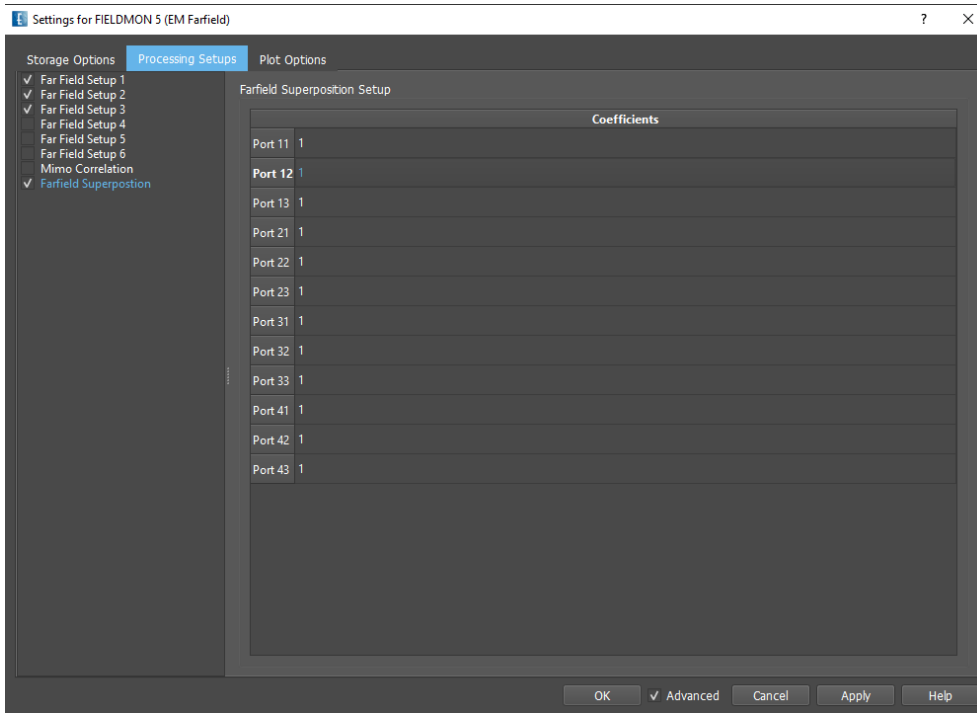


Figure 19.10.: Farfield Superposition Setup

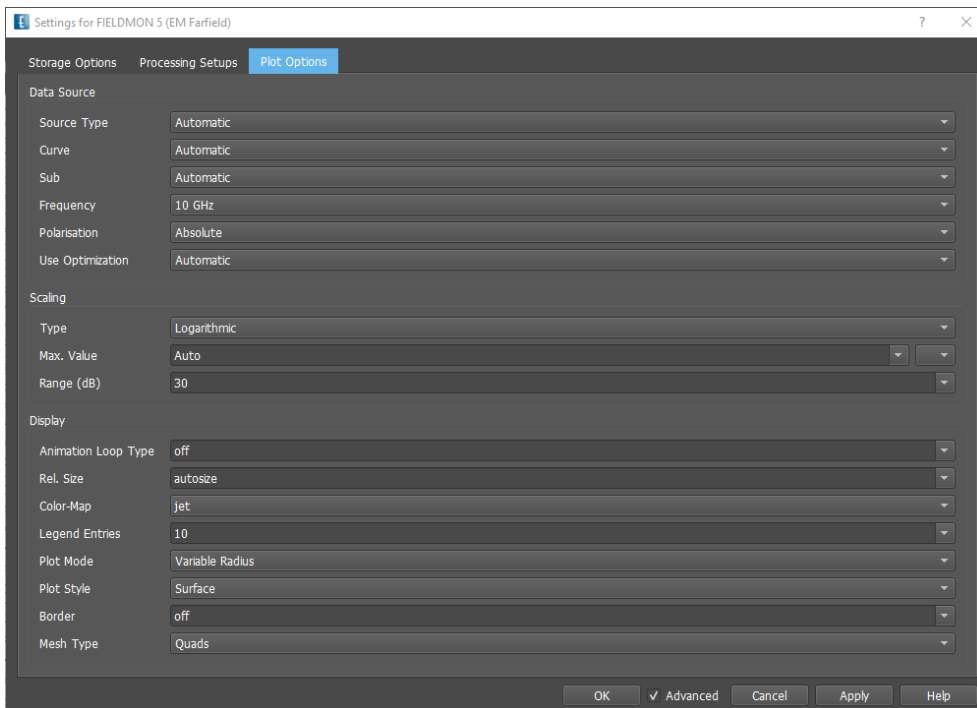



Figure 19.11.: Farfield Plot Options

- **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

### 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 polygonal or STL facets

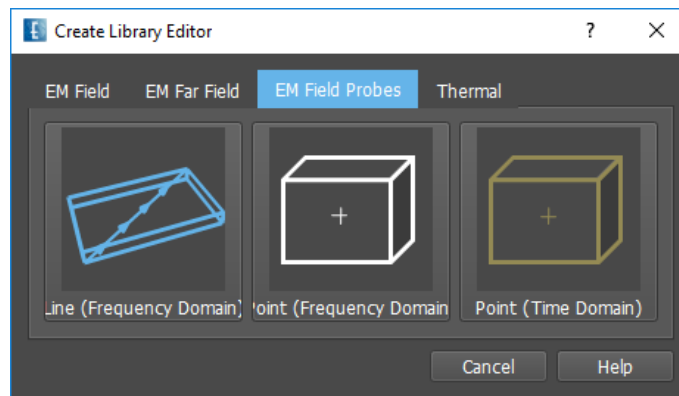


Figure 19.12.: Field Monitor EM Probes

## 19.4. 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

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
- **On-Disc Compression** Can be selected to reduce memory of output files. Higher compression rates may reduce accuracy.

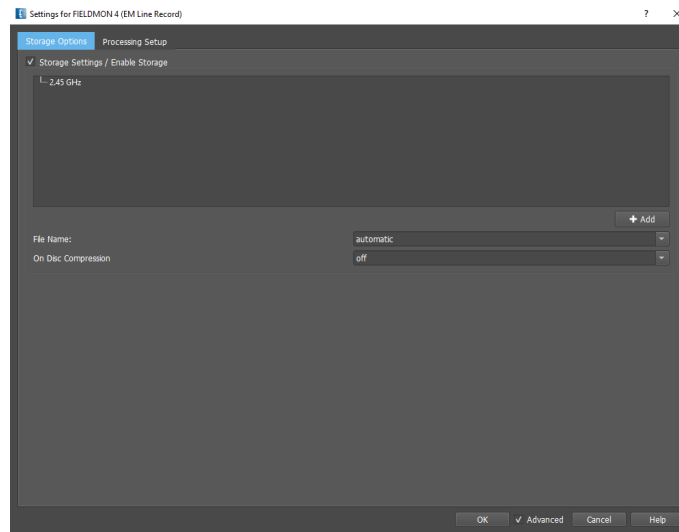


Figure 19.13.: Field line definition

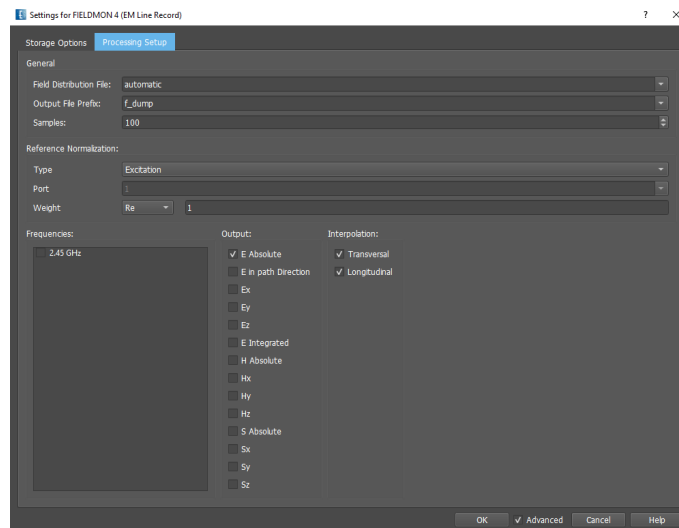


Figure 19.14.: Field line calculation

## 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)

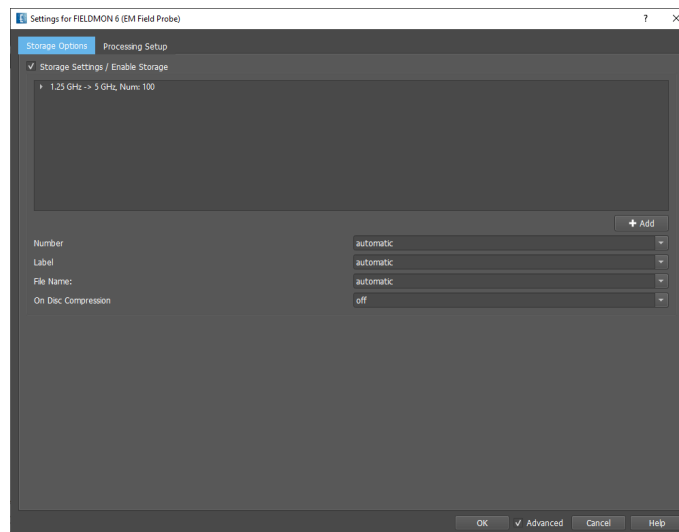


Figure 19.15.: Field probe definition

## 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
- **On-Disc Compression** Can be selected to reduce memory of output files. Higher compression rates may reduce accuracy.

## Processing Setup

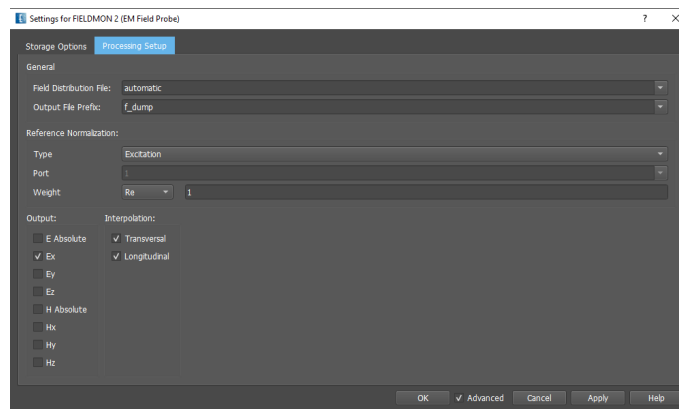


Figure 19.16.: Field probe processing setup

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

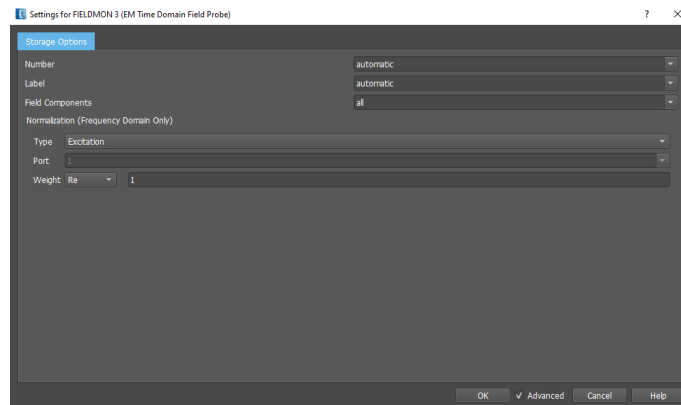


Figure 19.17.: Time Domain Field probe definition

### 19.4.3. Point Field Probe (Time Domain)

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.
- **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.

- **Data Source**
  - **Source Type** - If set to **Manual** the file name can be specified
  - **Sub** - Result folder selection if multiple ports have been used
  - **Use Optimization** - Browse variation folders



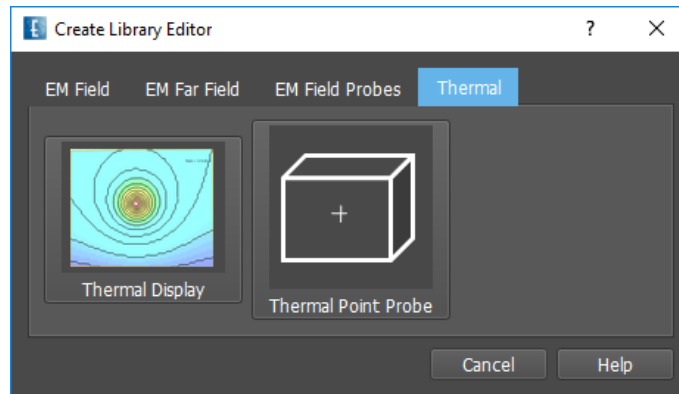


Figure 19.18.: Thermal Monitor

- **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, Fahrenheit) 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

### 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.

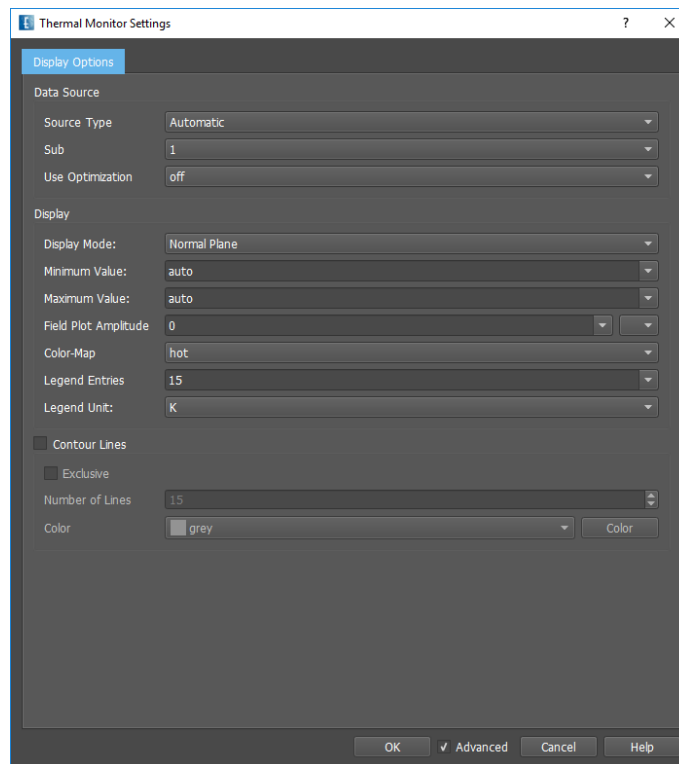



Figure 19.19.: Thermal Monitor Parameters

## 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 defined 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 definition 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 the right voltages or currents, the ports may not overlap with conductors because they could fully or partially short-circuit the port elements.

The current flowing out of the port with number  $n$  will be denoted as  $itn$  in time domain and  $ifn$  in frequency domain. The voltage at port  $n$  will be denoted as  $utn$  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  $n$  will be denoted as  $Zn.in$ .

## 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 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 accessed 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 not overlap.

- **Table Style**
  - **Detail View** The table is arranged with port parameters in columns. Sorting can be applied 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

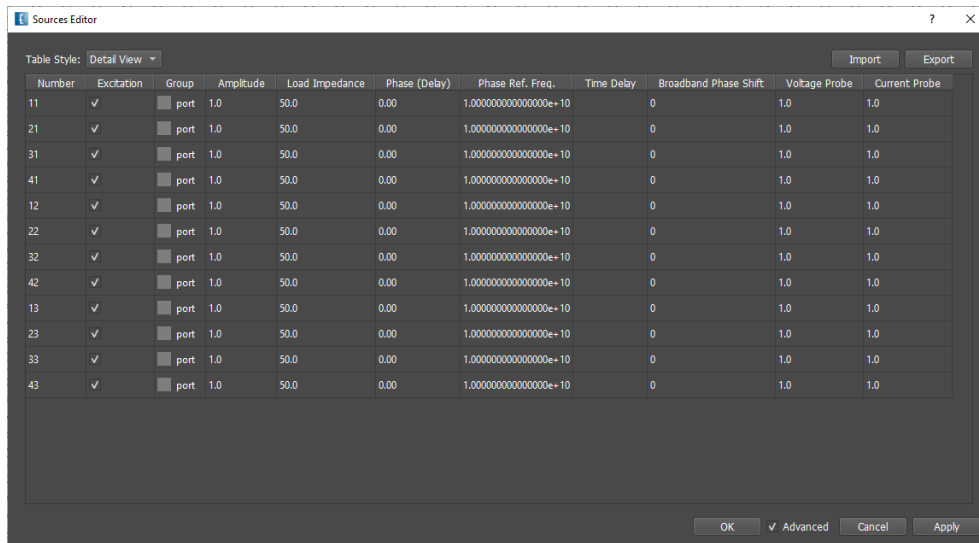


Figure 20.1.: Port Setup Wizard

- **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 Delay** Angle in degrees valid for a single frequency
- **Phase Ref. Frequency** Reference frequency used for phase delay
- **Time Delay** Value in seconds
- **Broadband Phase Shift** For AC simulations, the phase shift may be defined for a frequency range
- **Voltage Probe** Weight factor for voltage recording
- **Current Probe** Weight factor for current recording

#### Hint: Reference Impedance

The default reference impedance for separating incident and reflecting wave is  $50\Omega$  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 **On** in the port settings. In case of QTEM ports, this switch is set to **On** by default.

The actual used impedances are listed in the **Port Setup** of the **Simulation Setup** in the simulation tab.

#### 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.

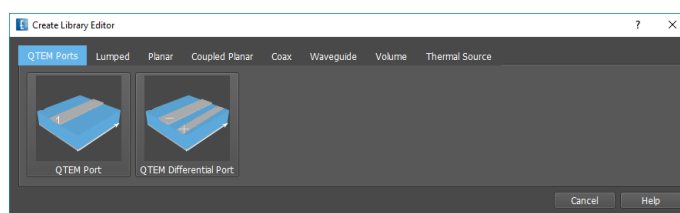


Figure 20.2.: QTEM port types

### 20.1.2. QTEM Ports

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)
- QTEM differential port (+,- or +,-,Ground system)

The signal line(s) are referenced by points on the line which determine start position and wave direction. 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
- **Point 1** determines the positive electrode and position of the port cross section
- **Point 2** determines direction of the wave (any point on the positive electrode)
- **Point 3** (Differential Port) determines the negative electrode
- **Object Editor** Points, name and group can be adjusted
- **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

### 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.

## 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  $\Omega$ , 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
  - **Frequency for port parameters in Hz** To be changed if different from **Target Frequency** defined in the **Simulation Setup**
  - **Tolerance  $\epsilon_{seff}$ ...** The static eigenmode is used as excitation if the effective permittivity difference of the dynamic mode is larger than this value

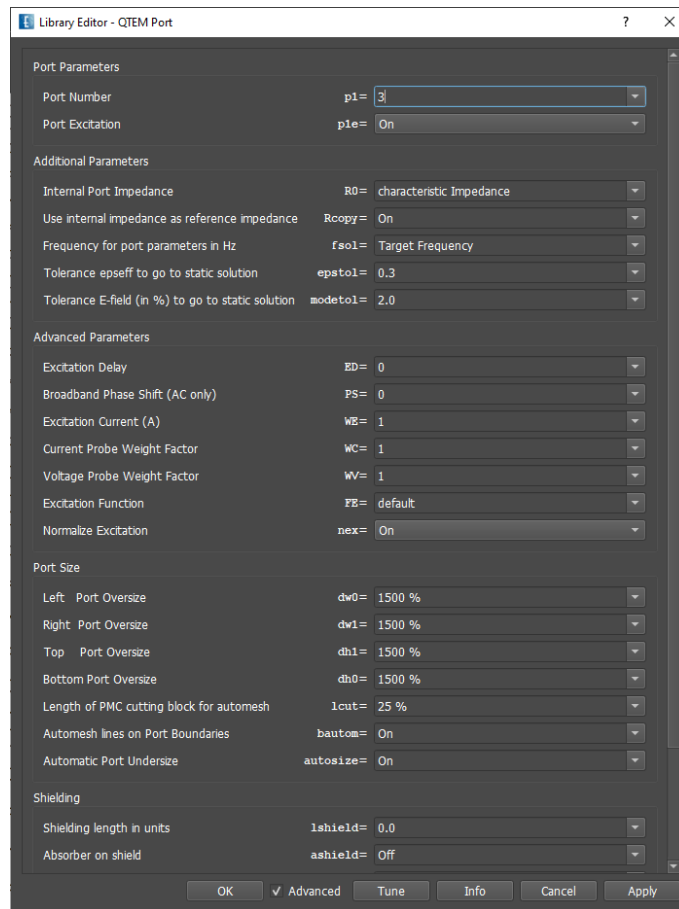


Figure 20.3.: Parameters of the QTEM Port



- 
- **Tolerance E-field ...** The static eigenmode is used as excitation if the electric field deviation of the dynamic eigenmode is larger than this value
  - **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
  - **Port Size** Determines the size of the line cross section to be analyzed. (100% refers to the distance between signal and ground)
    - **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
    - **Length of PMC...** Thickness of the magnetic wall which is placed behind the load and excitation plane
    - **Automesh lines...** If enabled, mesh lines are placed at the 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...** length of the absorber
    - **Absorber on shield** On or off
    - **Shield priority** Geometric priority in case of intersection with other objects
  - **Display**
    - **Drawing Mode**
    - **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  $50\Omega$ )
- 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  $\Omega$ , internal load of current source
  - **Use Impedance for postprocessing**  $Rcopy = On/Off$ . If set to **On** this value will be used for S-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 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

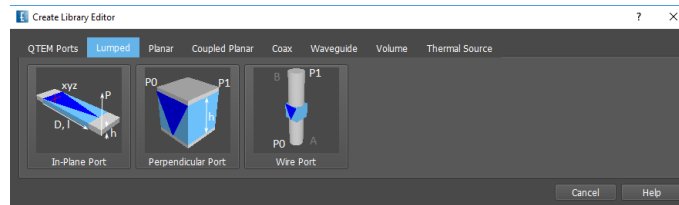


Figure 20.4.: Lumped port types

## In-Plane Port

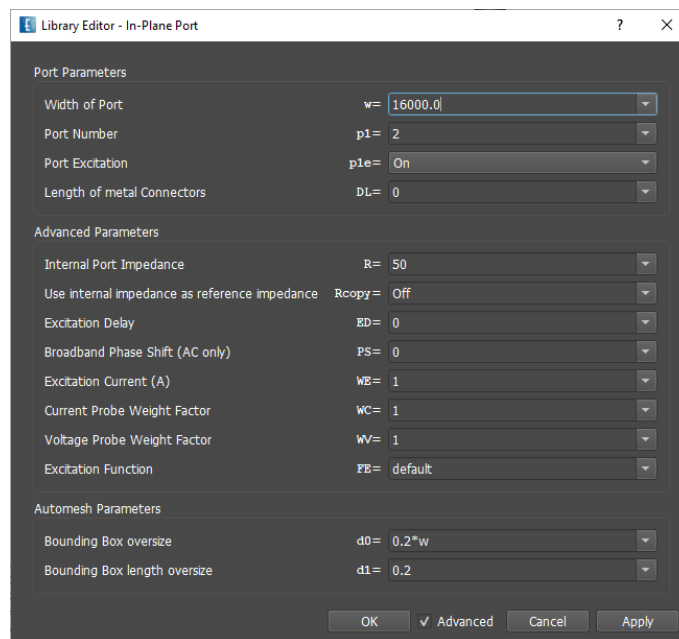



Figure 20.5.: Parameters of In-plane Port

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

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

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  $50\Omega$ )
- 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 a source area is simulated which contains the feed line geometry only
- If a source area is used the XPU algorithm can not be applied (slow simulation)

By default the phase reference plane of a feedline port is placed a certain distance apart from the excitation (to establish the QTEM mode). If this reference plane should be fixed to a location the port can be set to a calibrated port:

#### 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)

#### Common Port Parameters

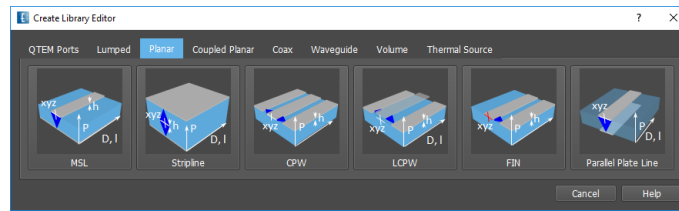


Figure 20.6.: Planar port types

- 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 port in units. Position of reference plane (for calibrated ports, only)
- **Measurement Distance**  $p1d > 1cell$  distance from base port 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  $\Omega$ , internal load of current source
- **Use Impedance for postprocessing**  $Rcopy = On/Off$ . If set to **On** this value will be used for S-Parameters
- **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. Permittivity**  $epsr \geq 1$  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 height 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 conductor
5. Point on inside of outer conductor

### 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

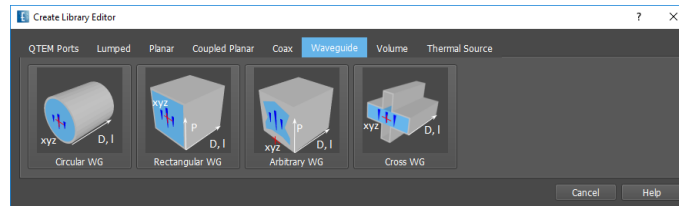


Figure 20.7.: Waveguide port types

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 excitation 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 character, 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

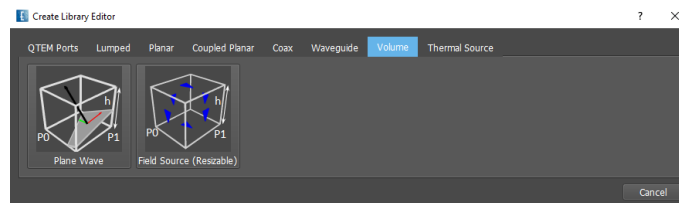


Figure 20.8.: Volume sources

### 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.

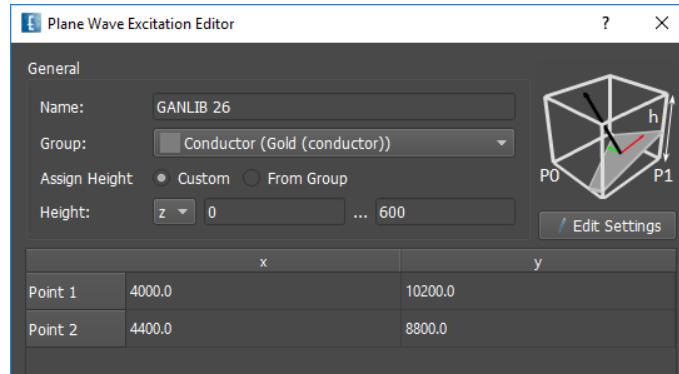


Figure 20.9.: Parameters of the plane wave

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:

direction	polarization	$\theta$	$\varphi$	$E_\theta$	$E_\varphi$
$\hat{x}$	$\hat{y}$	$90^\circ$	0	0	$E_0$
$\hat{x}$	$\hat{z}$	$90^\circ$	0	$-E_0$	0
$\hat{y}$	$\hat{x}$	$90^\circ$	$90^\circ$	0	$-E_0$
$\hat{y}$	$\hat{z}$	$90^\circ$	$90^\circ$	$-E_0$	0
$\hat{z}$	$\hat{x}$	0	0	$E_0$	0
$\hat{z}$	$\hat{y}$	0	$90^\circ$	$E_0$	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** -  $\theta$ - component for direction of propagation in **degrees**
  - **K.phi** -  $\phi$ - component for direction of propagation in **degrees**
- **Field Strength**
  - **E.theta** -  $\theta$ - component of electric field for polarization in  $\frac{V}{m}$
  - **E.phi** -  $\phi$ - component of electric field for polarization in  $\frac{V}{m}$
- **Options**

- **Enable Plane Wave**, can be switched off for special purposes
- **Delay** in **s**, the excitation may be delayed against other ports or sources
- **Disable Sides**, sides can be switched off for special purposes
- **Display Plane Position** in percent. Used for display of the plane wave symbol only.

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. H<sub>x</sub>,H<sub>y</sub>,H<sub>z</sub>. 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 descriptors

#### 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

## 20.3. Thermal Source

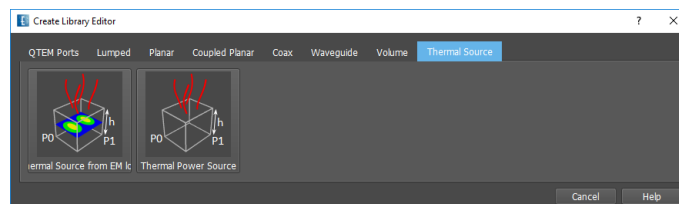


Figure 20.10.: Thermal sources

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
- **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

# 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. 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.

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**.

Further, 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

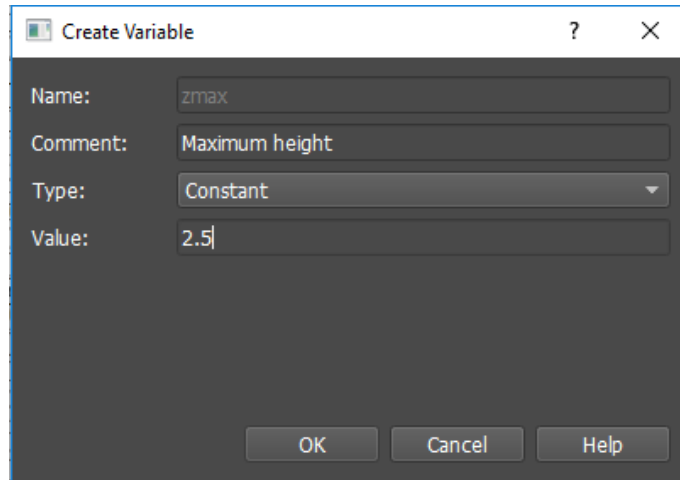


Figure 21.1.: EMPIRE XPU™ Variable editor

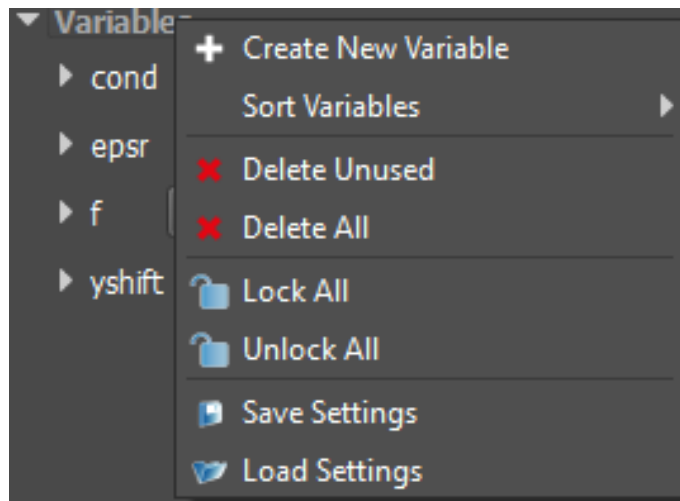




Figure 21.2.: EMPIRE XPU™ All variables control



## 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**
- **Load Settings:** Read file **.pms** and restore all name matching parameters

Right click on **Variable Name** opens the control of single variables

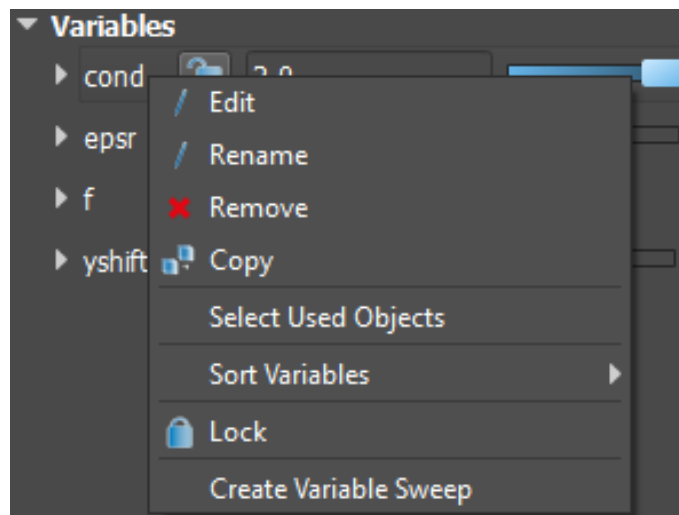


Figure 21.3.: EMPIRE XPU™ Single variable control

- **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
- **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	$pi$
$c_0$		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.

## 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 24.8 simulations in the simulation tab.

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.

## 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.


The mesh information **Mesh Info** is available in the navigation tree. 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.

### 22.1. Automatic Meshing

By default the automatic meshing is enabled and creates a suitable mesh for the objects and sources and entered 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.

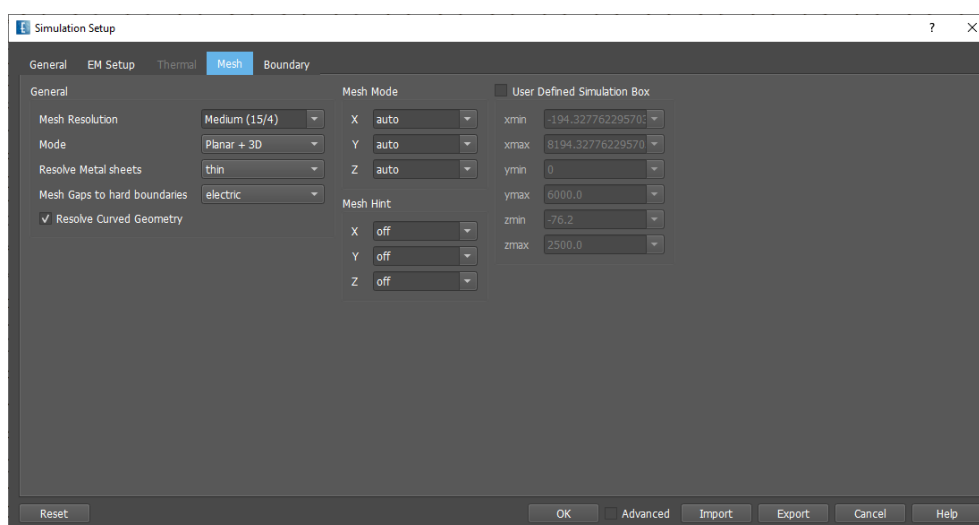



Figure 22.1.: General Mesh Adjustments

### 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 $N_\lambda$	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 number.

#### 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 neglect. 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

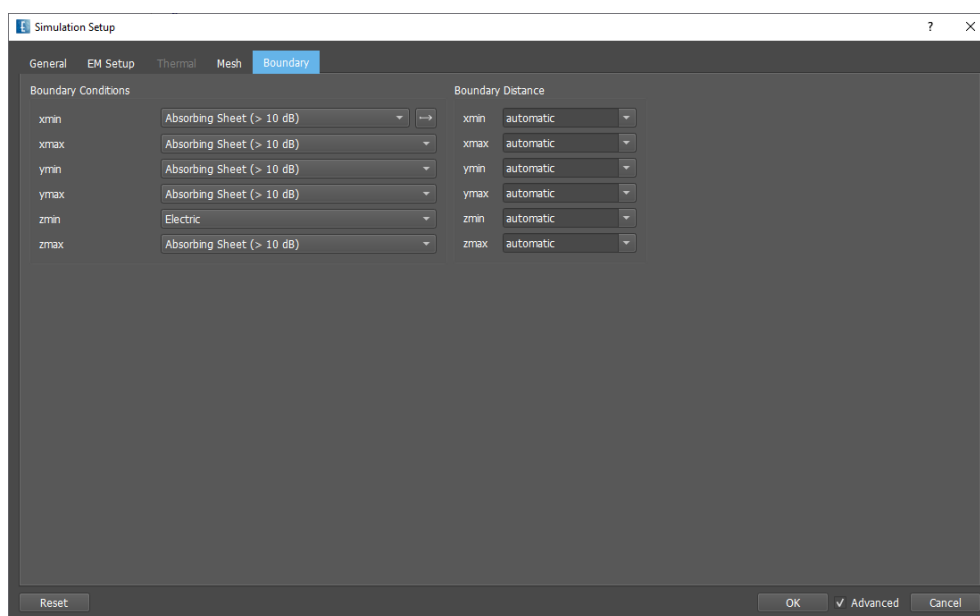


Figure 22.3.: Boundary Settings

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.



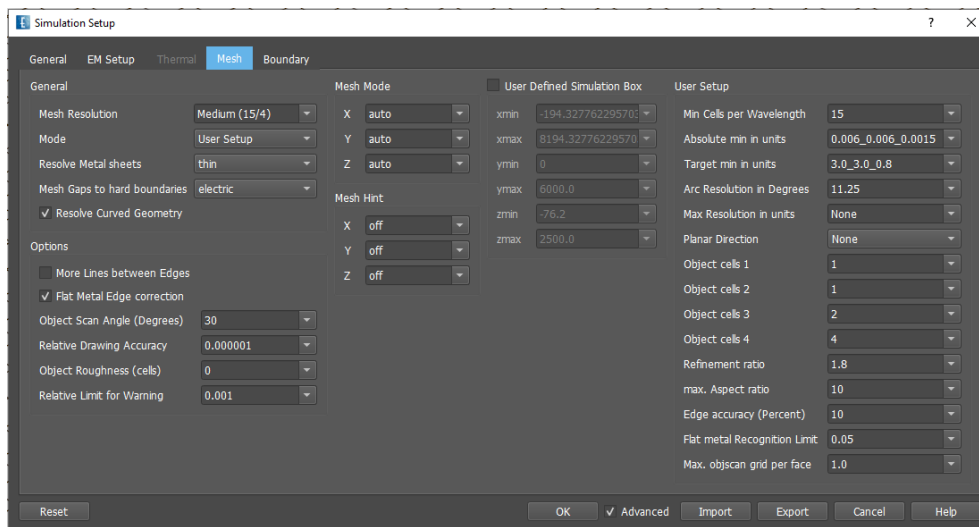


Figure 22.4.: Advanced Mesh Settings

### 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  $\lambda_{min}/N_\lambda$  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<sup>1</sup>, 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.

$$d_{min} = \min((x_{max} - x_{min})/intervals, \lambda_{min}/N_{\lambda})$$

$\lambda_{min}$  is calculated at the max. frequency (epsr = 1)

Resolution	<i>intervals</i>	$N_{\lambda}$
Coarse	1000	10
Medium	2000	15
Fine	4000	20

<sup>1</sup>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 resp. 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 only be used 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

### 22.2.1. Marking intervals




If mesh lines are already available intervals can be marked with a red dot by:

- left click between lines
- create a 1D arrow and click **Mark Range** 
- click **Mark All** 

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  $\delta$ : Add mesh lines in the marked intervals with the spacing of  $\delta$
-  **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** Save current settings in a file
- **Export** Load settings from a file
- **Cancel** Disregard current settings and close window
- **Help** Open the manual at the respective help page

### 23.1. General Setup

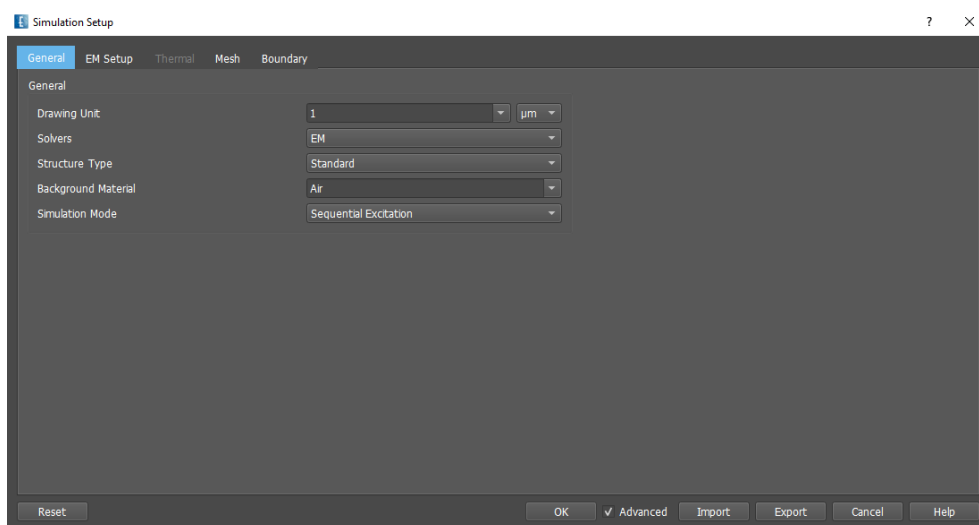


Figure 23.1.: Global simulation parameters

- **Drawing Unit** The drawing unit determines the relation between one unit in the drawing plane and the geometry in reality. The default is  $1\mu\text{m}$  and can be changed according to the entries in the list.
- **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
  - **Low Frequency Electric (Experimental)** Slow energy decay, low order estimation, high oversampling, high permittivity as background material
  - **Low Frequency Magnetic (Experimental)** Slow energy decay, low order estimation, high oversampling, high permeability as background material
  - **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 Extraction** 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

## 23.2. EM Setup

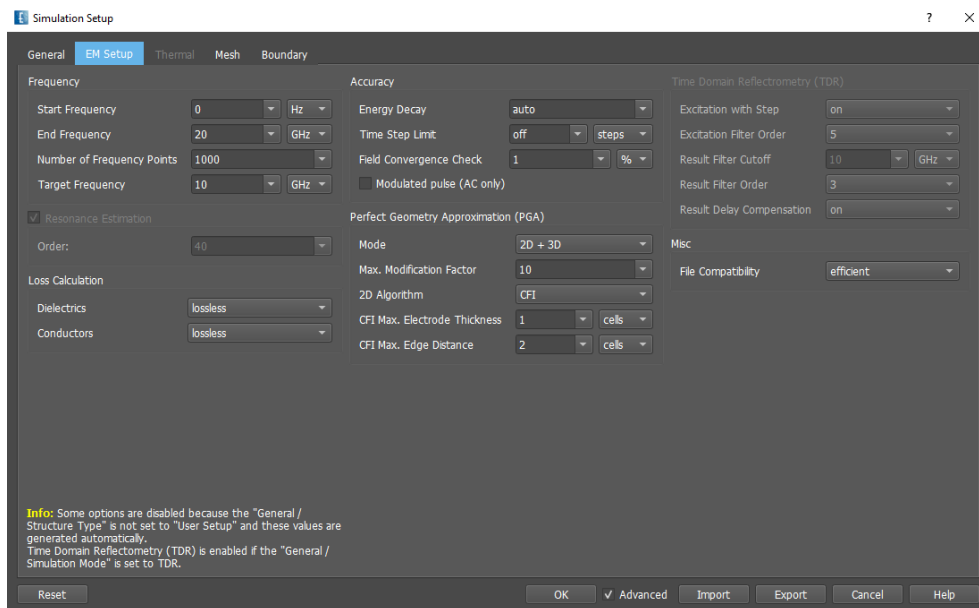


Figure 23.2.: EM simulation parameters

### 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.

#### 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).

### 23.2.3. Loss Calculation

By default **Dielectrics** and **Conductors** are treated as lossless. This reduces simulation time and leads to reliable results for parameters such as reflection or coupling. In case of resonators or if the transmission losses play a significant role and high accuracy is required the losses can be included in the simulation at different levels.

#### Dielectrics

For Dielectrics a loss tangent is assumed which should be constant over frequency. (Frequency dependent dielectrics can be treated with a general Debye material). 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, typically  $\pm 10\%$  bandwidth
- **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 modeled with the material property having a certain conductivity (bulk loss model). This model is recommended if the skin depth is in the range of the metal thickness. But this requires to resolve the skin depth which leads to a very fine meshing in this area and leads to long simulation time.

To avoid this, special surface loss models have been developed and used in EMPIRE XPU™ which do not require to resolve the skin depth so that planar metalization can be treated as flat and usually one mesh line is sufficient. 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
- **narrow band lossy**, constant conductivity, typically  $\pm 10\%$  bandwidth
- **medium band lossy**, 1st order approximation of skin effect typically  $\pm 50\%$  bandwidth
- **broad band lossy**, 2nd order approximation of skin effect, typically  $\pm 95\%$  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. 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

### 23.2.6. 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

### 23.2.7. 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

## 23.3. Thermal

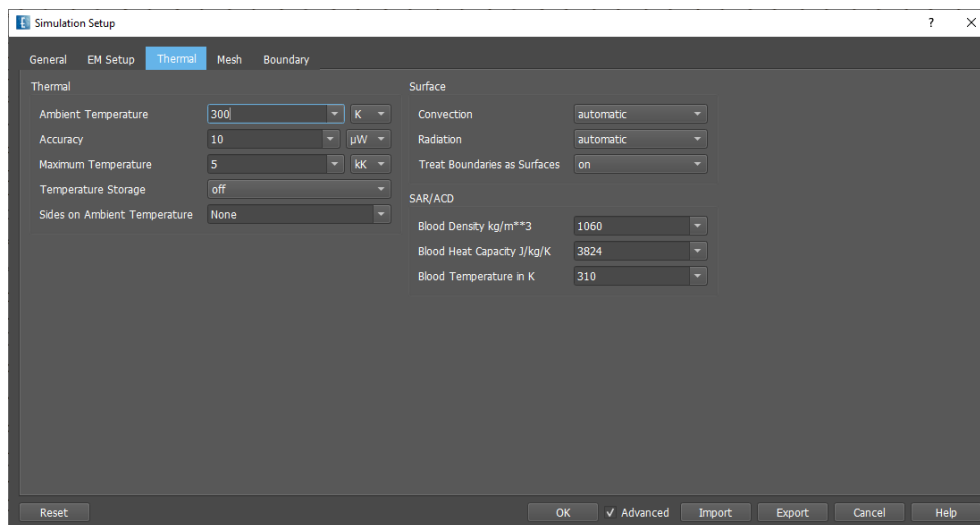


Figure 23.3.: Thermal simulation parameters

Only available if **Solver** is set to **Thermal**

### 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 ( $310\text{K} = 37^\circ\text{C} = 98^\circ\text{F}$ )

An additional body heat transfer mechanism can be set as **Basal Metabolism** in the respective object property as a thermal parameter.

## 23.4. Mesh

Mesh Setup as defined in chapter 22.1

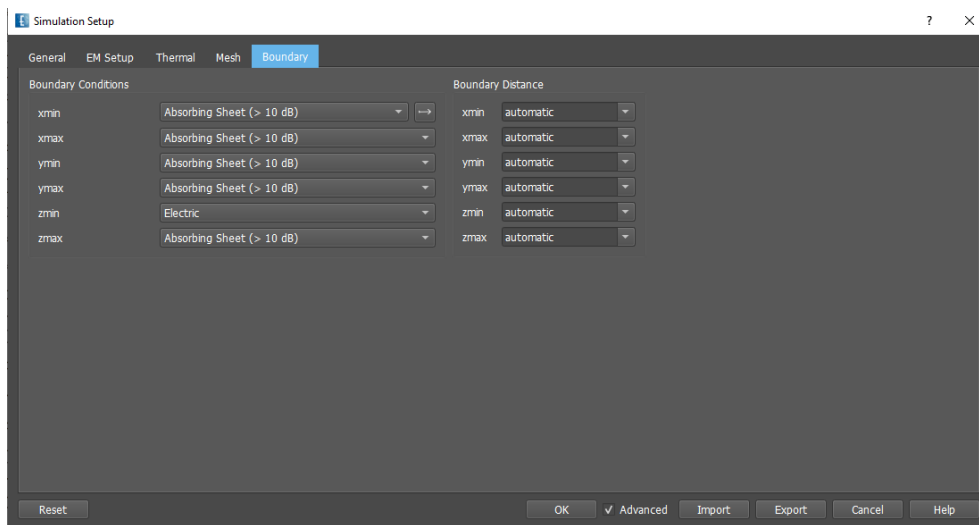



Figure 23.4.: Boundary setup

## 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.
- **Absorbing sheet** A sheet with (default)  $377\Omega$  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.

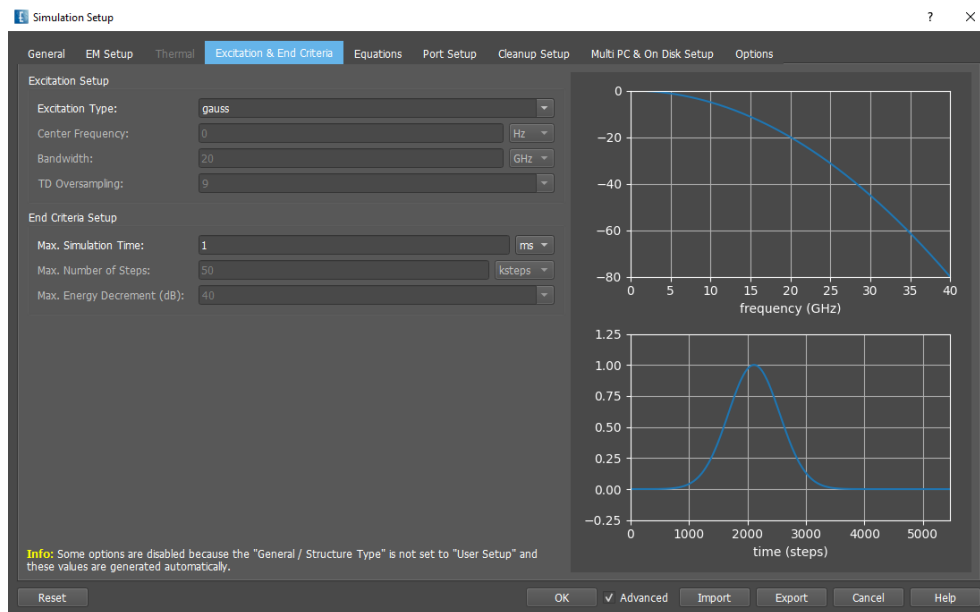


Figure 23.5.: Excitation setup

### 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:

$$e(t) = e^{-\left(\frac{t-t_0}{\tau}\right)^2} \sin(2\pi f_0(t-t_0))$$

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

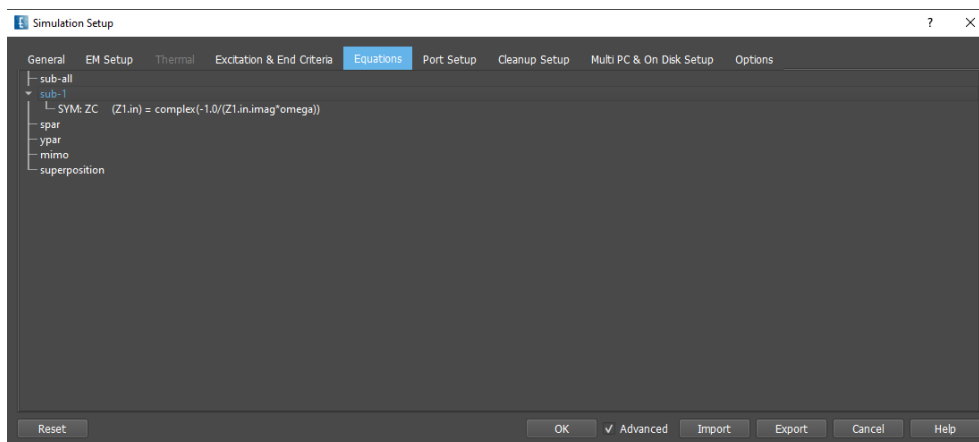


Figure 23.6.: Postprocessing Equation Setup

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.

$$y(f) = y_1(x_1(f)) + y_2(x_2(f)) + \dots$$

##### 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**  $2*\text{math.pi} * f$
- **.** 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).

$$y(t, f) = \text{folderexpression}/x(t, f)$$

#### 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 available 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

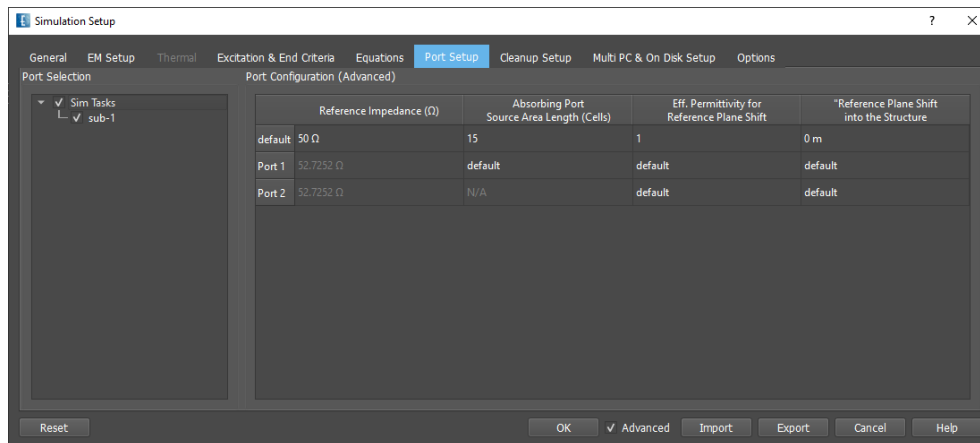


Figure 23.7.: Port setup

## 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  $\delta x_b$  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

### 23.10.1. Slave Host Setup

Here, the slaves for multi-PC simulation can be selected if the EMPIRE XPU™ 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**

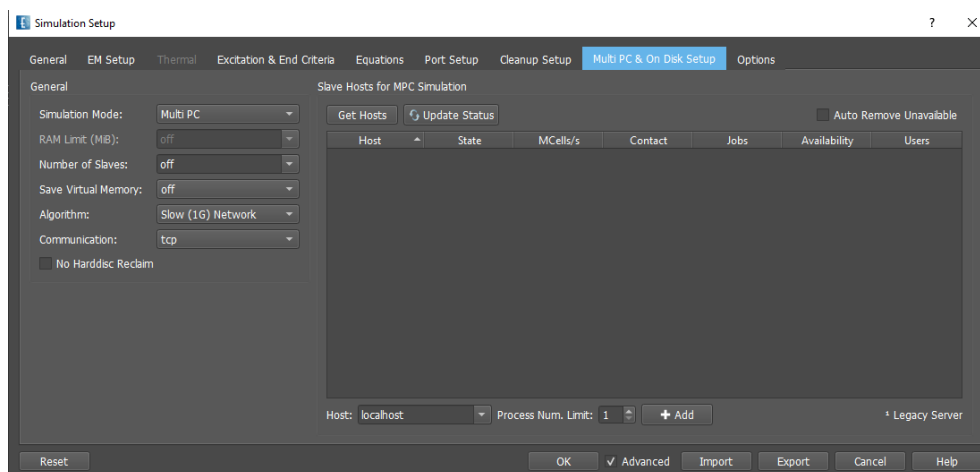


Figure 23.8.: Multi PC Solver setup

## 23.11. Simulation 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.

### 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.

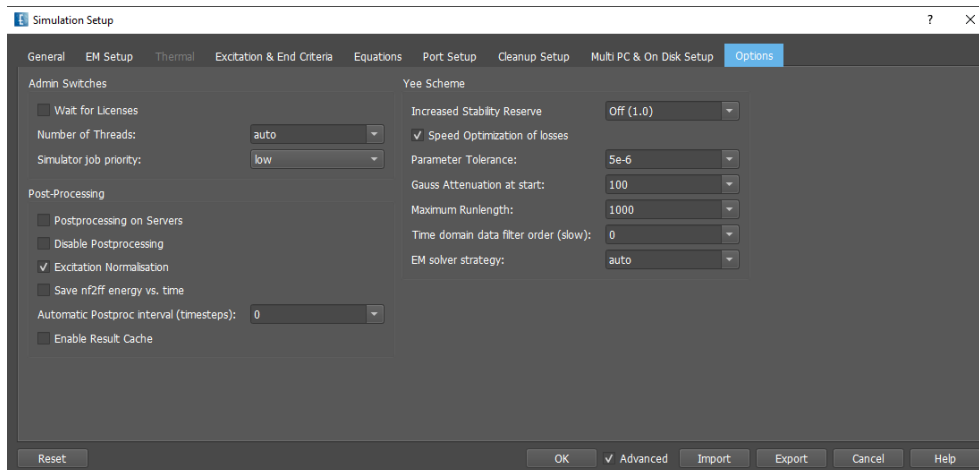


Figure 23.9.: Simulation Options

- **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

#### 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 files have to be available in the project folder.

### 24.2. Quick Start

A Simulation Control Window is opened by pressing the **Start Simulation**  button from one of the Design tabs.

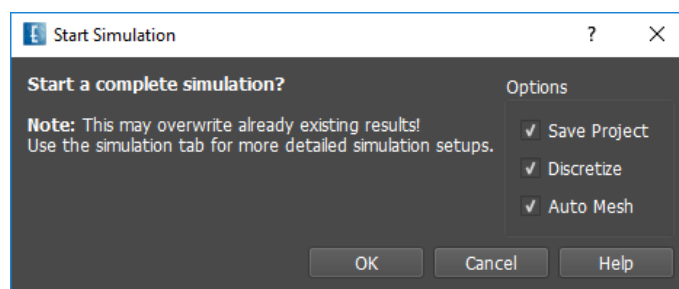


Figure 24.1.: Simulation Start Window

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.

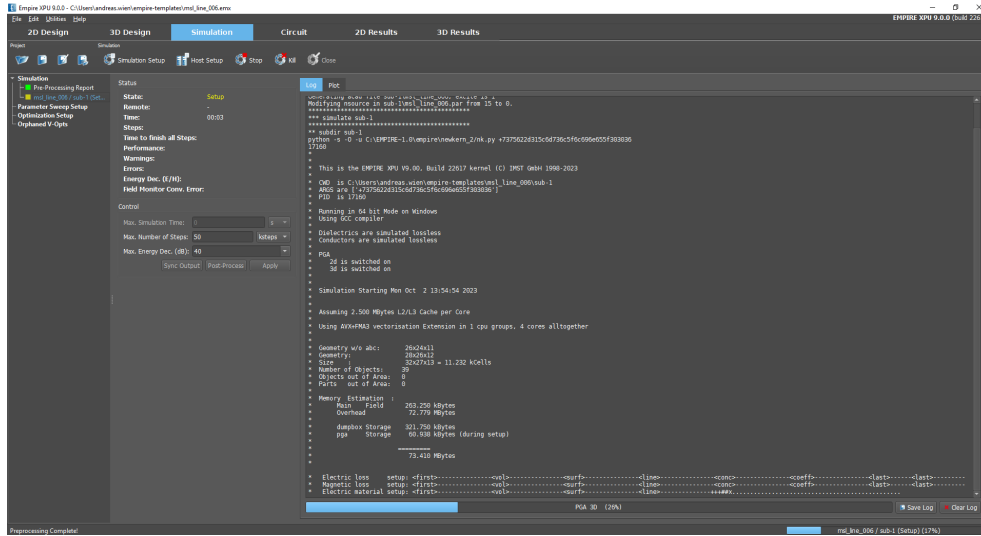


Figure 24.2.: Simulation Log Window

### 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 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 <sup>1</sup>
- **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 preprocessing

During the simulation the log window shows:

- **Multistep Strategy** Optimum 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

## 24.4. Basic Simulation

The simulation can be started by clicking the **Start Simulation** button  in one of the **Design** Tabs 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.

<sup>1</sup>Estimation without PGA or QTEM ports

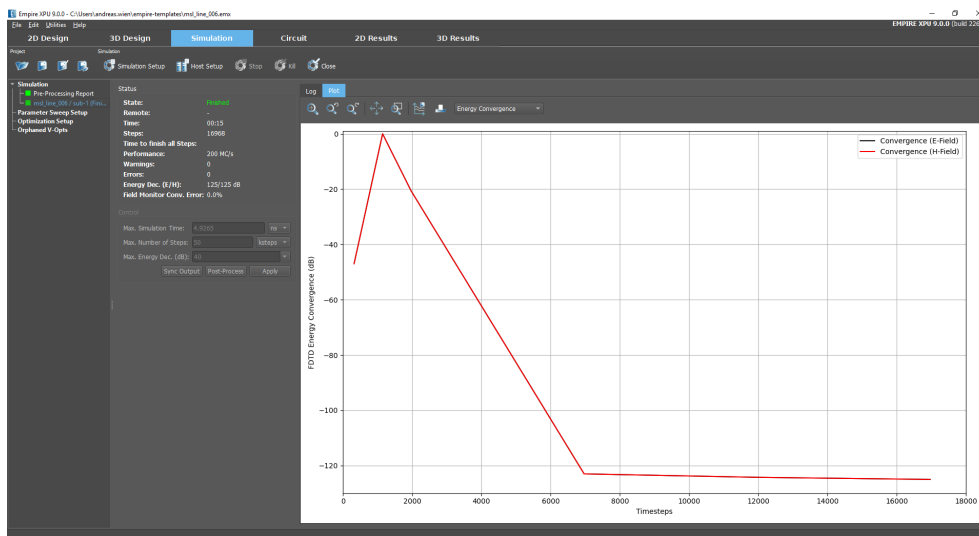




Figure 24.3.: Simulation Plot Window

## 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.

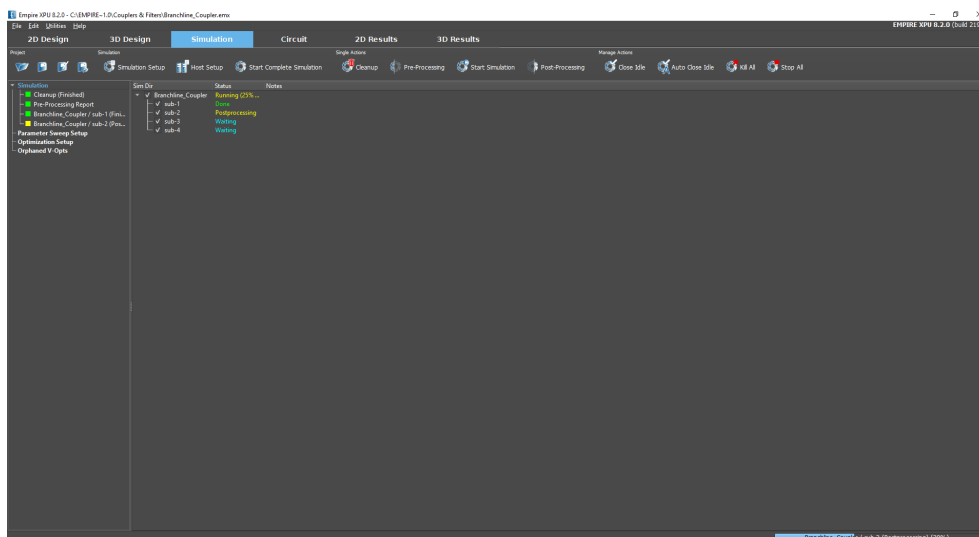


Figure 24.4.: Snapshot of Simulation

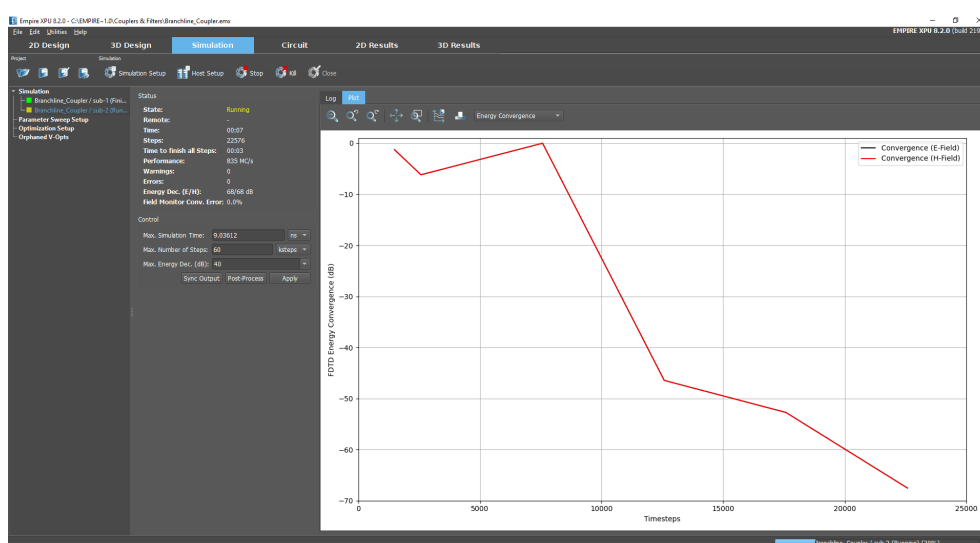


Figure 24.5.: Snapshot of a single job

Available buttons for Simulation (Select **Simulation** 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
- **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 nearfield data to get intermediate results for near and far fields
- **Post-Process** Terminate current simulation, create dump files and start postprocessing
- **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.

## 24.6. Batch processing

If several jobs should be queued they can be added with right click on **Simulation**, 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

In order to invoke other PCs see chapter 24.9

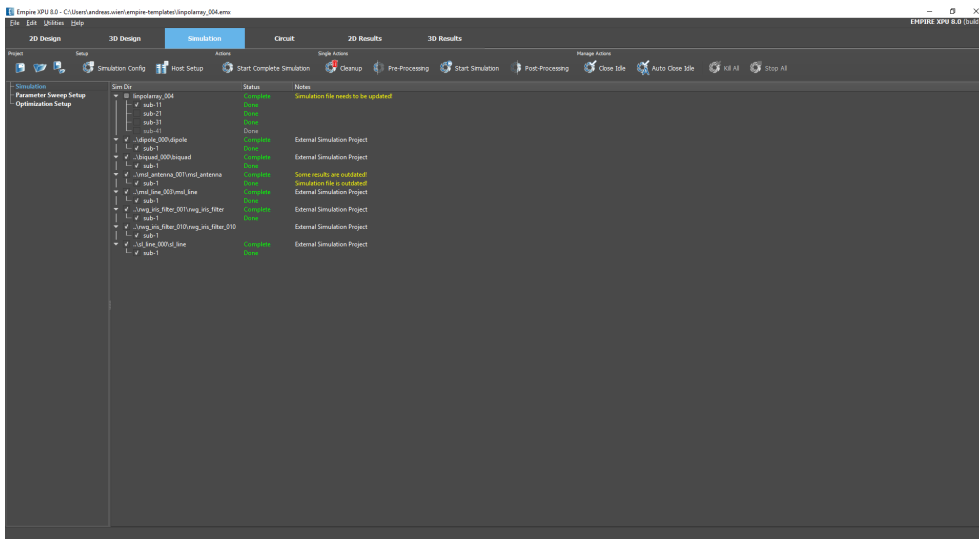


Figure 24.6.: Batch control window

## 24.7. Parameter Sweep

If variables have been defined, they can 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.

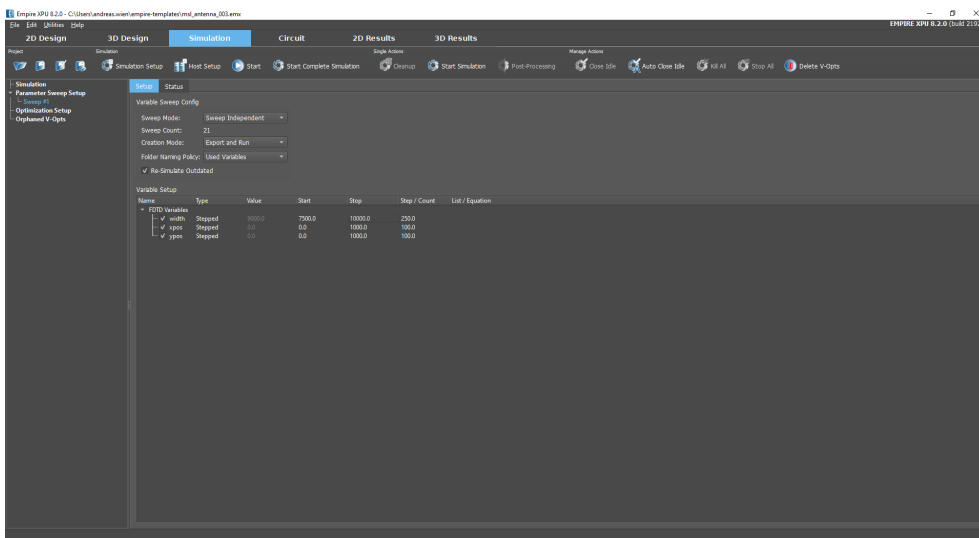



Figure 24.7.: Parameter sweep control window









### 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.

## 24.8. 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.

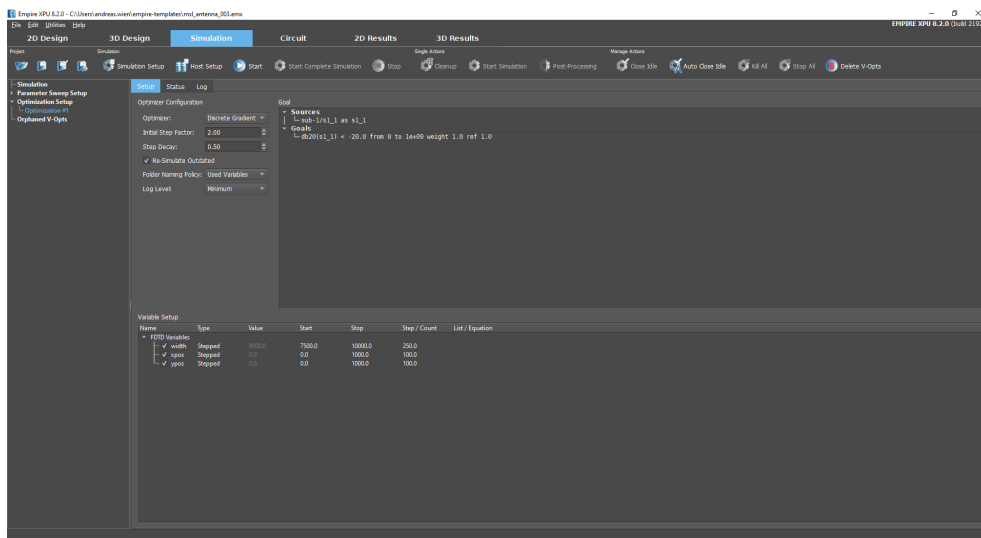











Figure 24.8.: Optimizer control window

### 24.8.1. 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



## 24.8.2. 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 – occasionally even for noisy behavior. There are theoretical situations where it 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.
  - **MLSLS** - 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 satisfies 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.

### 24.8.3. 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, ...)<sup>2</sup>

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** definition 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.

<sup>2</sup>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**

**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:

$$E = \sqrt{\frac{\sum_{i=1}^m \frac{weight_i}{n} \sum_{k=1}^n \left( \frac{expression_{i,k} - goal_i}{ref_i} \right)^2}{\sum_{i=1}^m weight_i}}$$

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. 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** has been started on them, they can be used to run the simulation remotely. This works for Linux and Windows and cross installations.

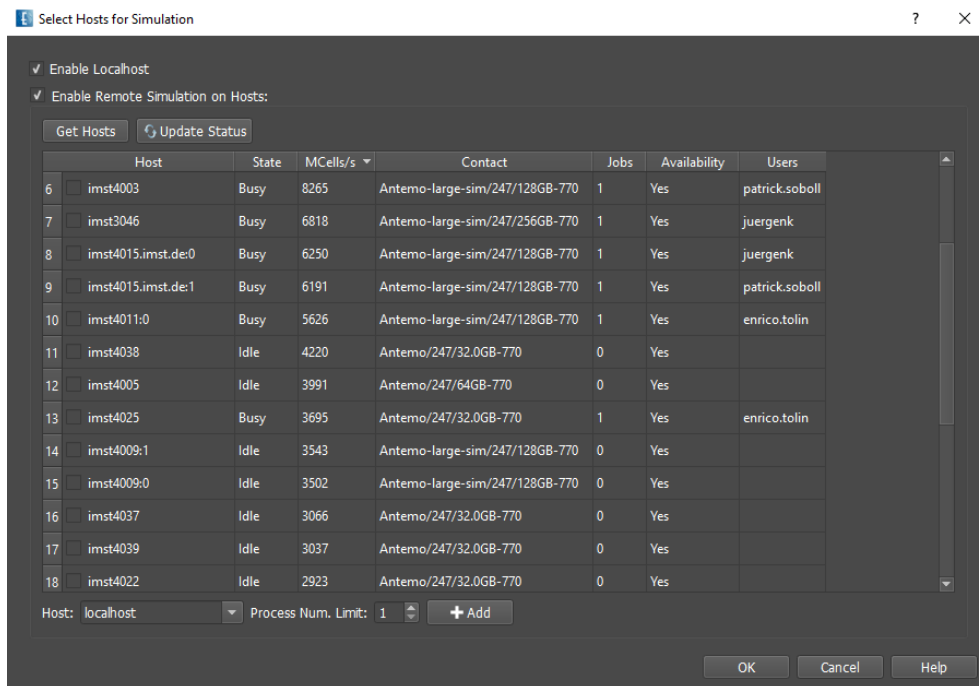


Figure 24.9.: Hosts control window

- **Get hosts** - Get a list of all available hosts in the LAN
- **Update Status** - Refresh list of available host


- **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
- **Host** - Hostname in network
- **State** - **busy** if host is processing an own 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

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.

## 24.10. 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  $\varphi$  refers to the angle between x axis and y axis and  $\theta$  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

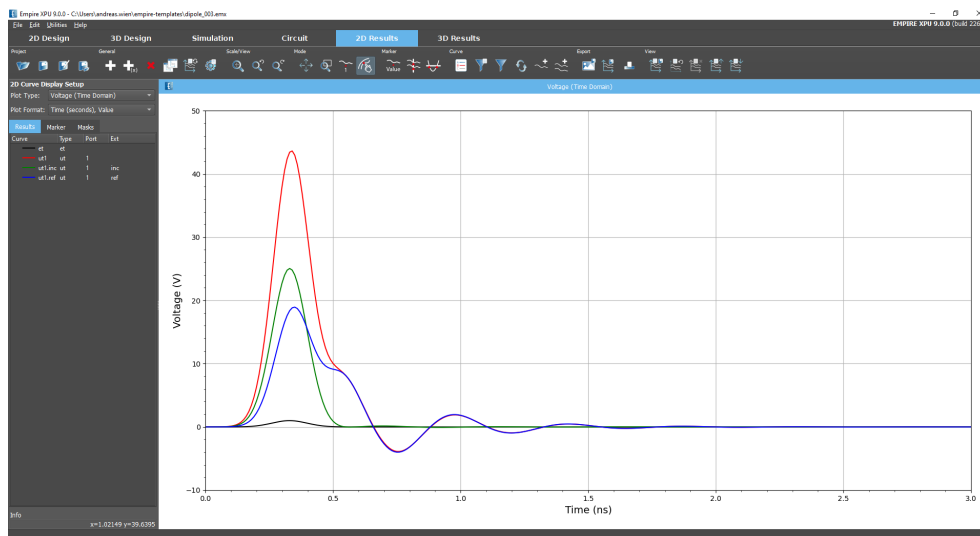






















Figure 25.1.: 2D graph display for voltage

### 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. Because 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

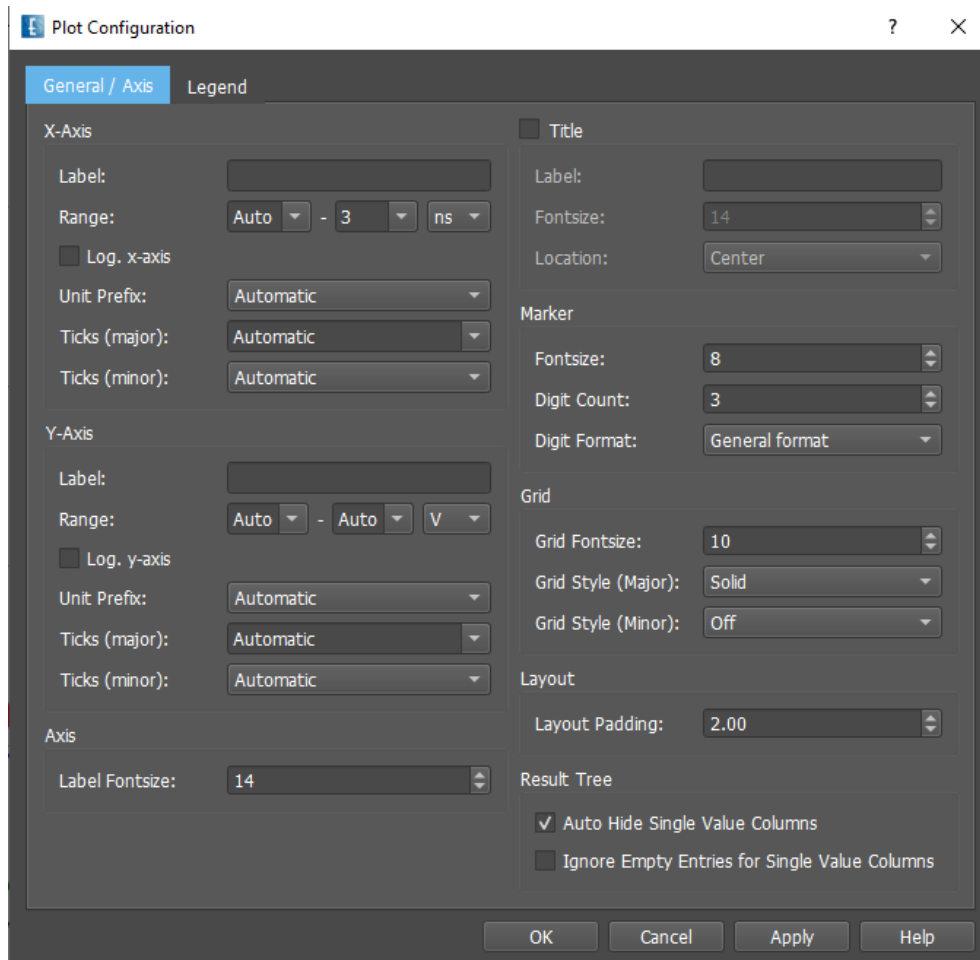


Figure 25.2.: Global plot settings (right click in plot area)

### 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 Reflectometry 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
- $\mu_0$ \***H-Probe (Frequency Domain)** Magnetic flux over frequency
- $\mu_0$ \***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
- $\mu_0$ \***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 Curve, custom(f)** user defined frequency domain results. 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** =  $\frac{1}{2\pi} \frac{d\varphi}{df}$
- **Group Delay (el.length)** =  $\frac{1}{2\pi c_0} \frac{d\varphi}{df}$
- **Angle Unwrapped (deg)** 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  $50\Omega$
- **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.

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

### 25.1.6. 2D Curve Display Setup - Marker

Markers can be created with the marker buttons 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)
- **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

### 25.1.7. 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 25.3
- **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

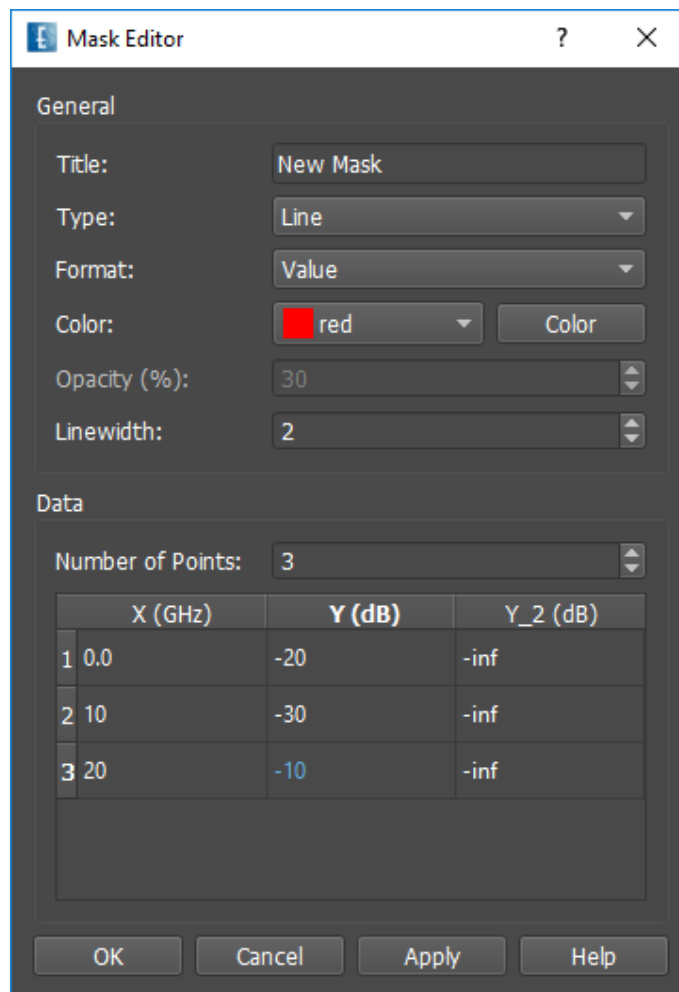




Figure 25.3.: Mask Editor

### 25.1.8. 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_ecom` 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.

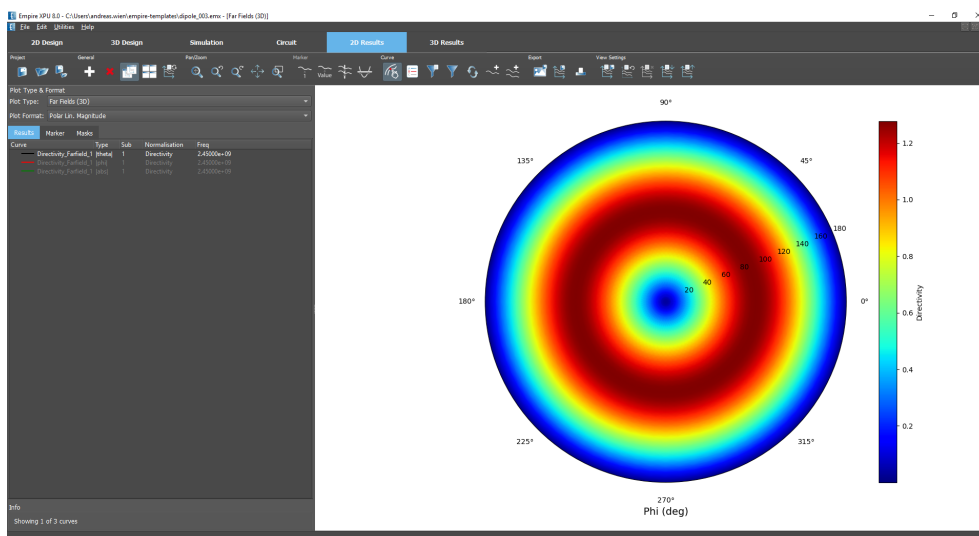


Figure 25.4.: Color map of a far field

## 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 excited 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 cannot 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 transformation 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	Port k defined
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	
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

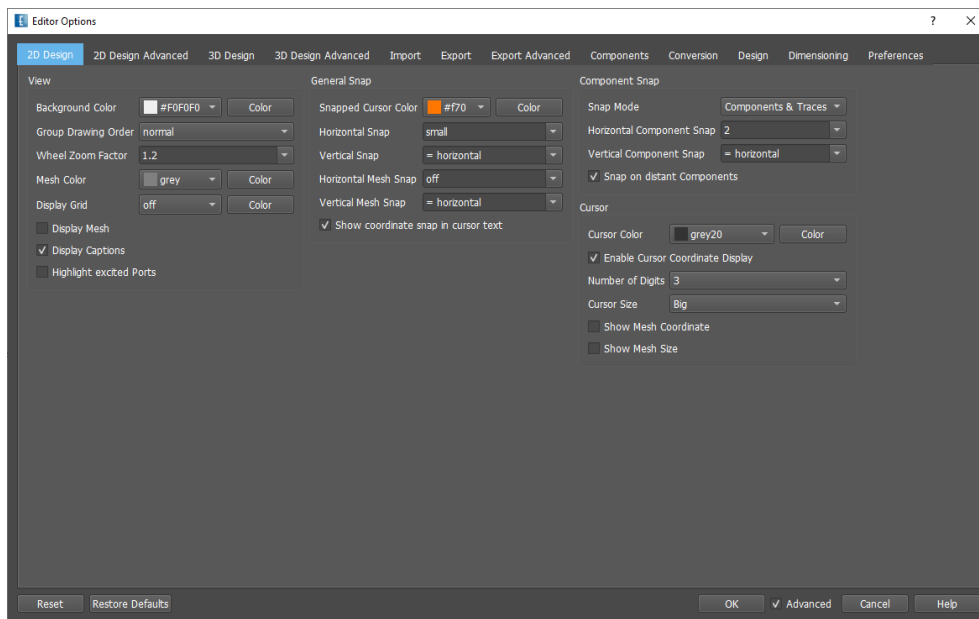



Figure 26.1.: Option menu

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 . 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
- **Wheelmouse 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.

### 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 enabled, 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
- **Number 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 continuous 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

- **EXCELLON decimal digits** Number of digits used in EXCELLON file
- **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
- **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



### 3D Solids

- **Facets (STL/OFF/Nastran)** Polygonal mesh Representation
  - **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 Representation
  - **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 and their specific options can be set.

## 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 algorithm for adjacent 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  $\phi_0$**  Angle at start point
- **Slope Angle  $\phi_1$**  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
- **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
- **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
  - **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

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

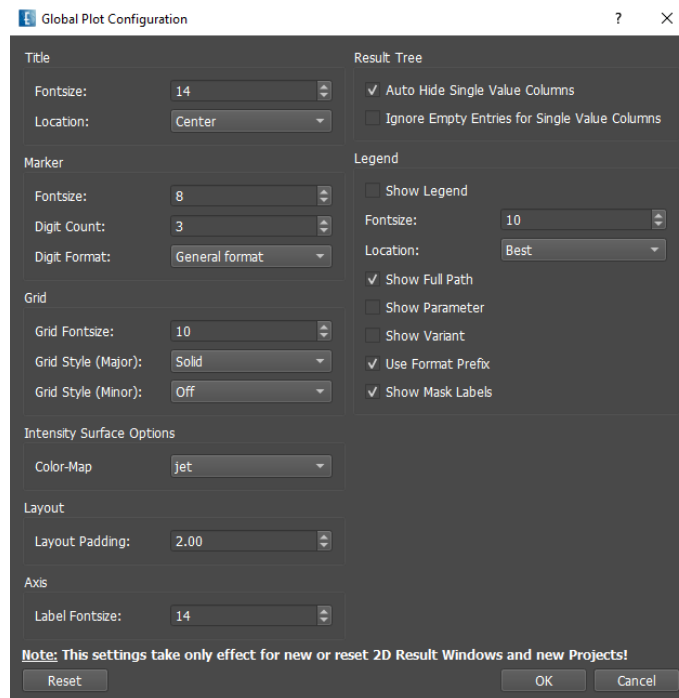


Figure 26.2.: Option menu for 2D Results

- **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
- **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.

### 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 of markers and arrow
- **Show Position** Display coordinates if enabled



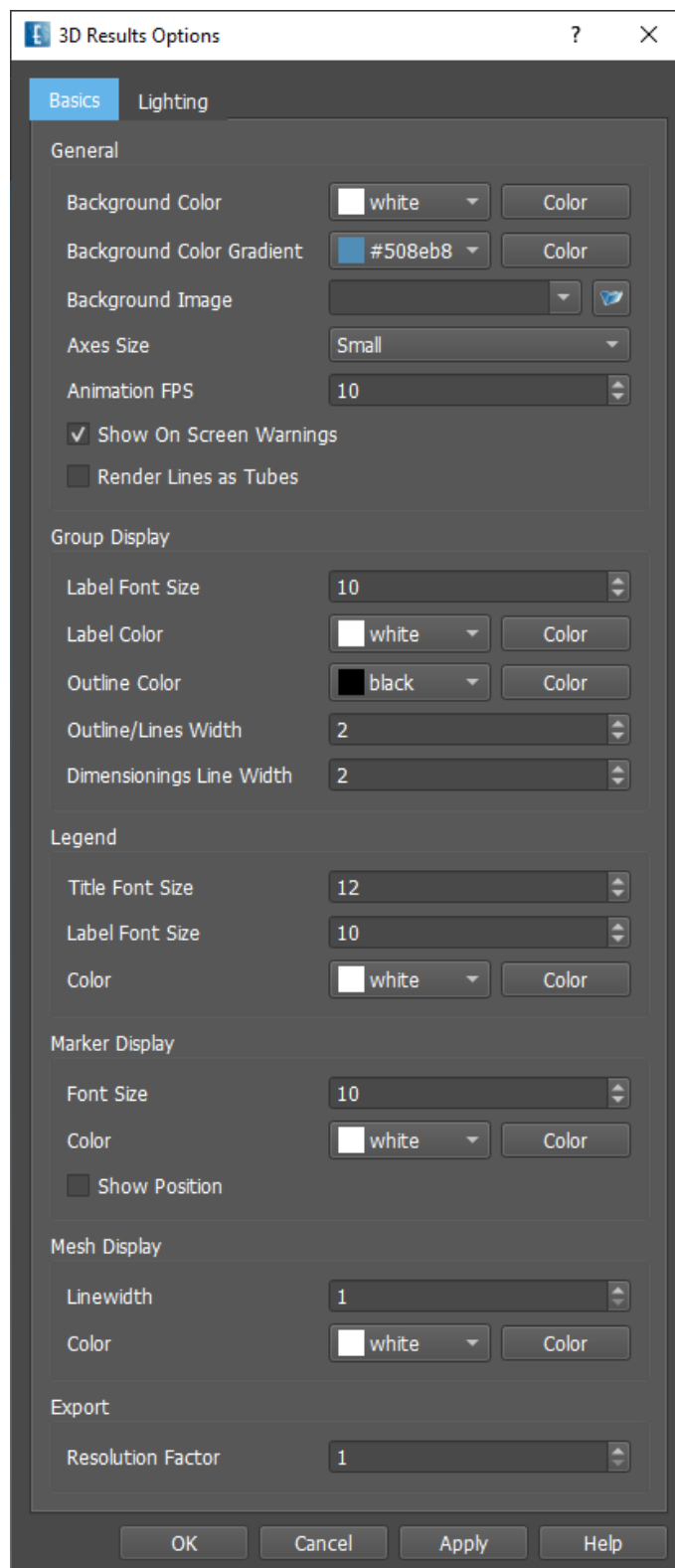




Figure 26.3.: Option menu for 3D Results

- **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. 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  $\theta$ ,  $\varphi$  (moves with camera)
- **S** Scene Light with Intensity and direction  $\theta$ ,  $\varphi$  (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**.

## 27. Project Wizard

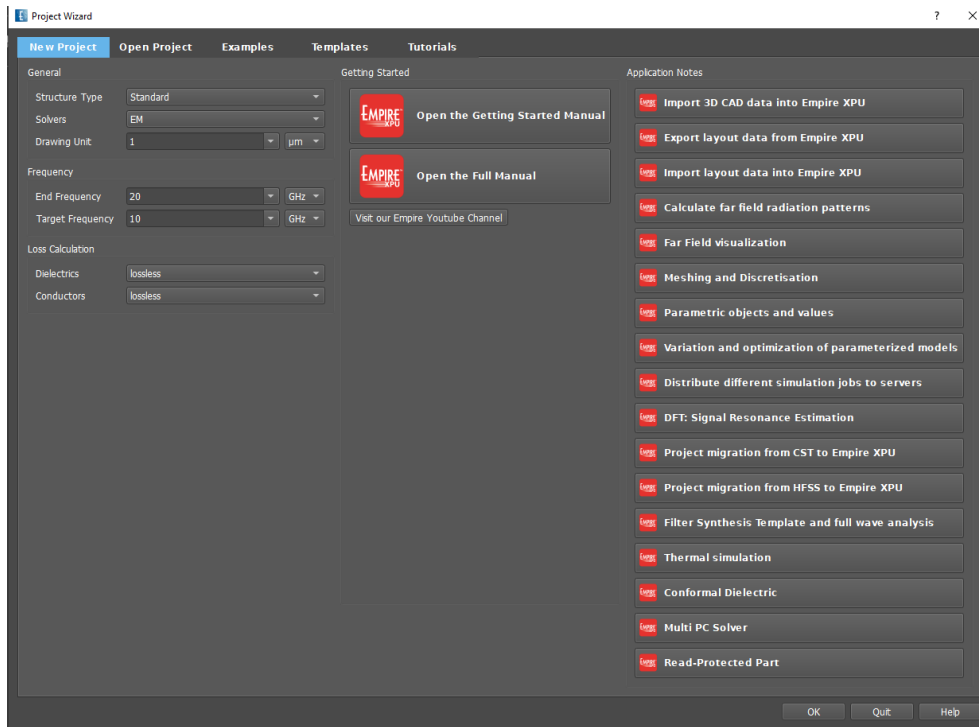


Figure 27.1.: Project Wizard: New Project

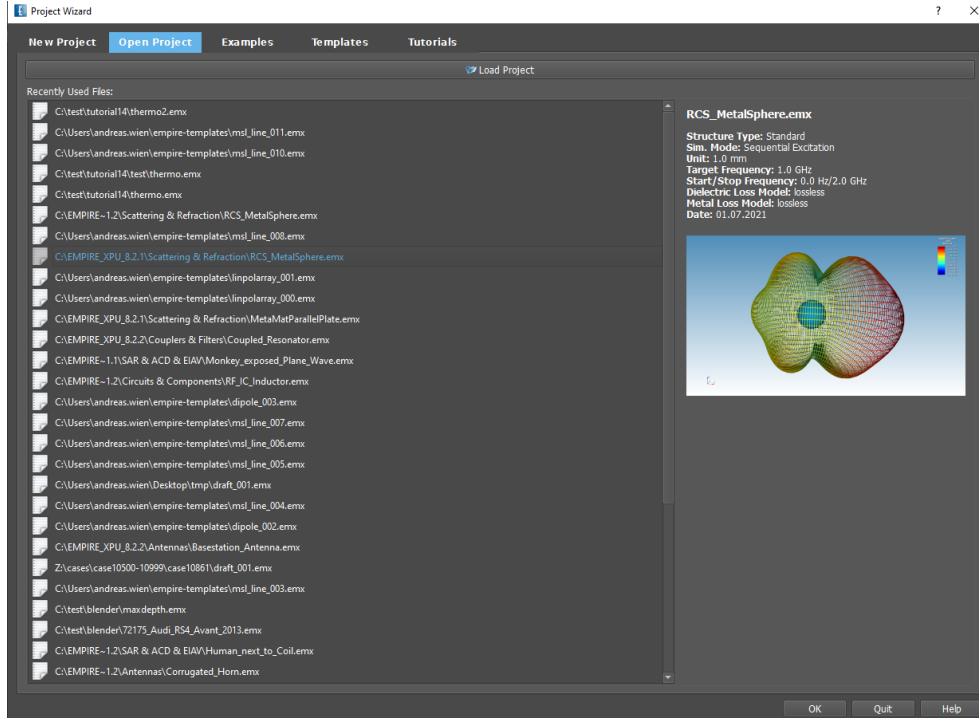


Figure 27.2.: Project Wizard: Open Project


## 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

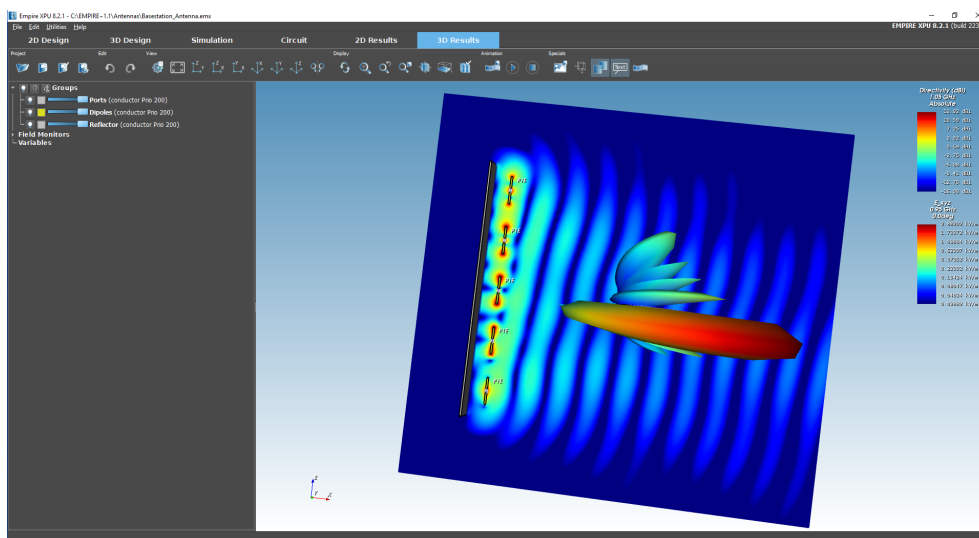




Figure 27.3.: Base station example

## 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 design frequency

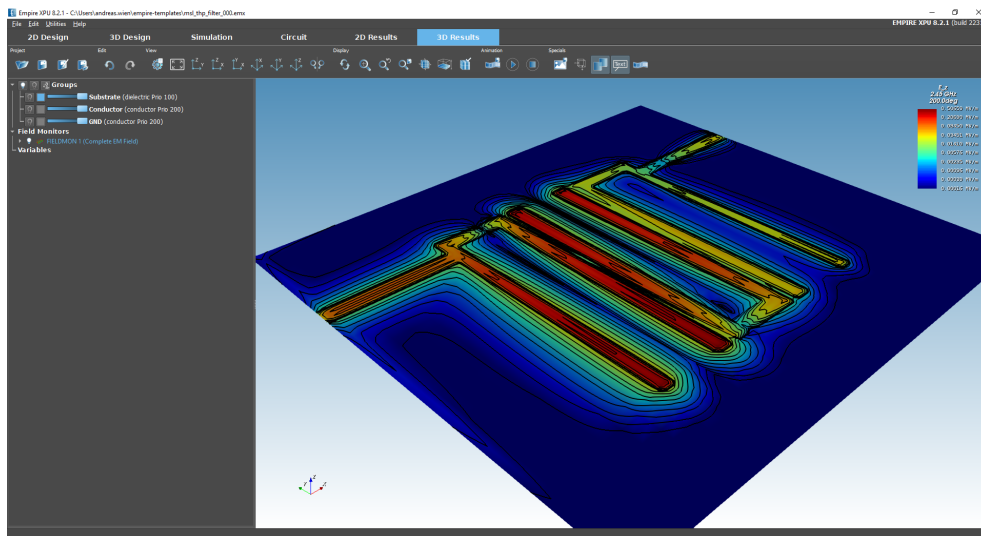


Figure 27.4.: Filter Template

### 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 ground 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.

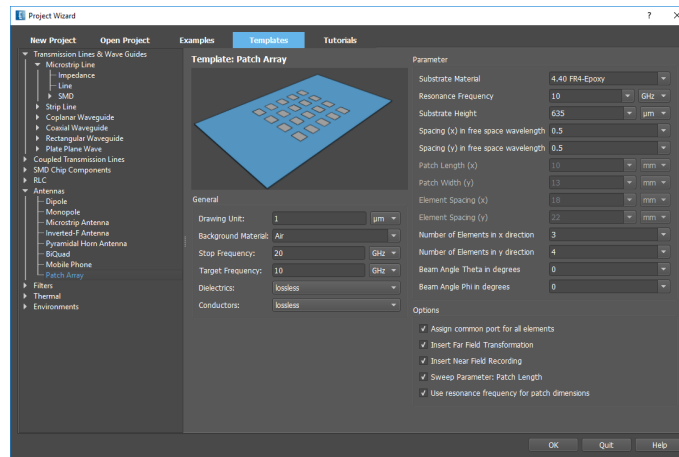


Figure 27.5.: Patch array template

### 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  $\mu\text{m}$ ) 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  $\mu\text{m}$ ) 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 rectangular 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  $40\text{mm} < w < 100\text{mm}$  and  $70\text{mm} < h < 150\text{mm}$ . 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 ( $\lambda$ ) 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  $\lambda$ . 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 ( $\theta$ ) and phi ( $\varphi$ ). 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.

### 27.9.1. Coupled Lines

The layout contains straight quarter wavelength coupling sections. The coupling distance is determined from the even and odd impedances which are derived from 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

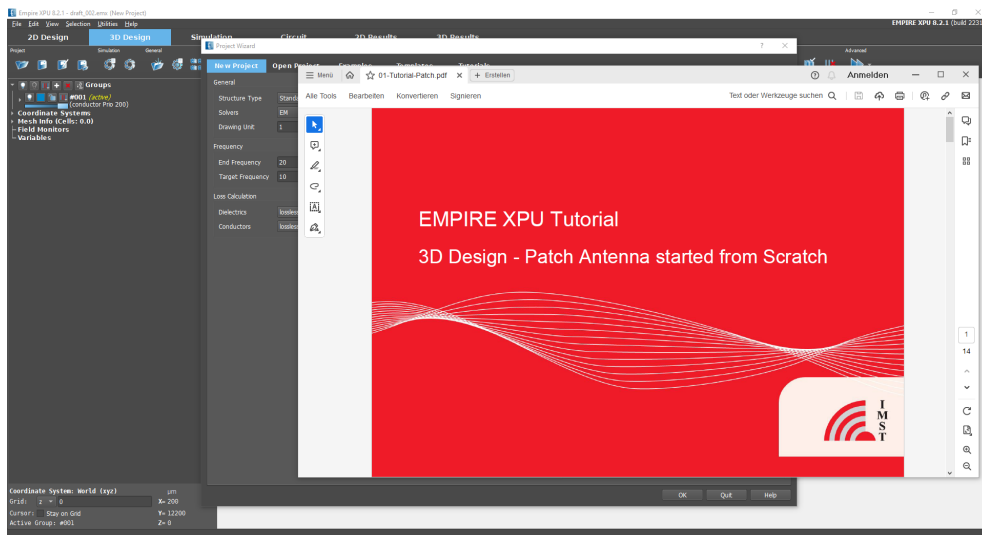


Figure 27.6.: Patch Antenna Tutorial

## 28. Scripting

EMPIRE XPU™ supports an interface to Python scripts. The **Python** programming language, commands and syntax is available at [www.python.org](http://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:

<code>from math import *</code>	Include libraries
<code>from numeric import *</code>	
<code>def anyname (pts,args):</code>	function call from <b>userpoly name</b> in EMPIRE XPU™
<code>p2d = []</code>	Initializing a 2 dimensional point list
<code>(x0,y0),(x1,y1) = pts[:2]</code>	Assigning points from a 2-point polygon
<code>Amp = eval(args[0])</code>	Every time the drawing is updated this value is renewed
 <code>LX = int(x1-x0)</code>	Interval size
 <code>for x in arange(0,LX,LX/100.):</code>	Loop over x values
<code>y = Amp *(x/LX)**2</code>	function with parameters
<code>p2d.append((x+x0,y0+y))</code>	Append points to the list
 <code>p2d.append((x1,y0))</code>	Append last point
<code>p2d.append((x0,y0))</code>	Append first point
 <code>return p2d</code>	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



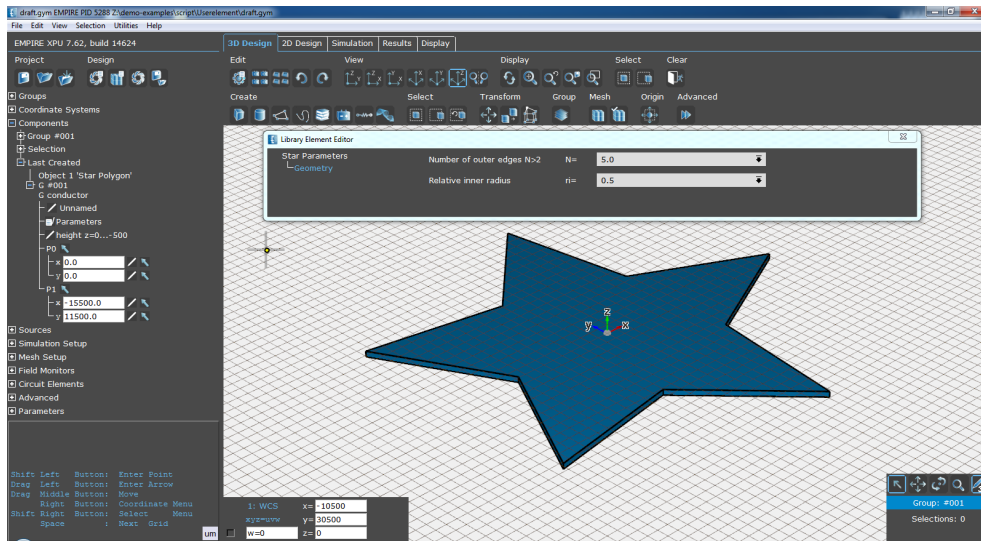


Figure 28.1.: Userdefined Element example

As an example, the script could be defined as follows:

<pre>def sigma(s):     x = s.X0 + s.X     if x &lt; 0 or s.component != 2: return 0     else: return x / 7000.0</pre>	<p>Define conductivity as a function of s</p> <p>Size of the material box in x-direction</p> <p>zero for points outside and x, y components</p> <p>Assigning conductivity for z component</p>
<pre>def epsr(s):     return 1.0</pre>	<p>Define permittivity as a function of s</p> <p>Set to 1 if not used</p>

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

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.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 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. `msh_line.py`) located at `C:\EMPIRE_XPU_8.xx\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_8.xx\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" "-0" "-u" "C:\XPU\env.py" script draft.emx script.py
```

Example for a run.sh file (Linux):

```
$HOME/XPU/PY3k-64/bin/python -0 -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"))

```

## 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 recommended way for script-development
- From the command line<sup>1</sup> or from another tool with the commands listed in 28.4

### 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

<sup>1</sup>Start cmd.exe from Windows. Copy the command from the empire-startup icon and adjust the command

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 regularly !!!
```
  
- `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 tessellated 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 rounded 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
- `layersselect(s,name):`  
select all objects on layer
- `layersubtract(s,name):`  
subtract 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_excite=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





**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

The following section describe specific settings for different applications.

## 29.1. 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 29.16, covering a volume large enough for averaging, minimum one cell thickness
- Adjust display parameters, Section 29.14, 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  $1\text{cm}^2$  averaging). The number can be edited.


### Examples

- Load **Examples - SAR & ACD - Disc in homogeneous Magnetic field**

## 29.2. 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

## 29.3. Capacitance calculation

### Usage

- Create a lumped port (number 1), Section 29.22, 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  $C = \text{Im}(Z_{1,\text{in}}) * (2\pi f)$

- Valid for low frequency (linear range of the  $1/\text{Im}(Z_{11})$ )

## 29.4. Circular Polarization (Antennas, Waveguides)

### 29.4.1. Narrow band

#### Usage


- Create an exciting port (number 1), Section 29.22, 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

### 29.4.2. Broad band

#### Usage

- Create an exciting port (number 1), Section 29.22, 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)

## 29.5. Efficiency calculation (Antennas)

### Usage

- Create a Far field recording, Section 29.8
- 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)

## 29.6. Far field animation

### Usage

- Create far field monitor before simulation, Section 29.8
- 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**

## 29.7. Far field plot

### Usage

- Create Far Field Monitor before simulation, Section 29.8
- 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**

## 29.8. 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 29.7, or Far field animation, Section 29.6

### 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**

## 29.9. Gain calculation (Antennas)

### Usage


- Create a Far field recording, Section 29.8
- 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

## 29.10. Impedance calculation

### Usage

- Create Port(s), Section 29.22
- 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  $Z_n.in$  and  $Z_k$  are available after postprocessing



### Examples

- Load **Tutorials - Multi Layer Spiral Inductor**



## 29.11. Inductance calculation

### Usage

- Create a lumped port (number 1), Section 29.22, 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  $L = \text{Im}(Z_{1,\text{in}}) / (2\pi f)$
- Valid for low frequency (linear range of the  $\text{Im}(Z_{1,\text{in}})$ )

### Examples

- Load **Tutorials - Multi Layer Spiral Inductor**

## 29.12. 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

## 29.13. Multiple port excitation (independent)

### Usage

- Create Port(s), Section 29.22
- 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**

## 29.14. Near field animation

### Usage

- Create near field monitor before simulation, Section 29.16
- 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**

## 29.15. 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

## 29.16. 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 29.14, or Near field plot, Section 29.15

### Remarks




- Recording of volumes requires the memory  $M = 48N_xN_yN_zN_f$
- After simulation a file named emvolume.dbx is generated (binary file)

### Examples

- Load **Tutorials - Low Pass Microstrip Filter**

## 29.17. Optimization

### Usage

- Define variables for geometry, physical or functional properties
- Check variable range and step
- **Simulation** Tab - **Optimization Setup**, Section 24.8
- 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**

## 29.18. 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**

## 29.19. Phase center calculation (Antennas)

### Usage

- Create a Far field monitor, Section 29.8
- 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

## 29.20. 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**

## 29.21. 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**

## 29.22. 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**

## 29.23. Quality factor calculation (Inductance)

### Usage

- Create a lumped port (number 1), Section 29.22, 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  $Q = \text{Re}(Z_{1.in}) / \text{Im}(Z_{1.in})$

### Examples

- Load **Tutorials - Multi Layer Spiral Inductor**

## 29.24. 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 29.20
  - the source area should fully enclose the scattering object
  - specify the incident wave direction (in spherical coordinates)
  - choose polarization (in spherical components)
  - 1 V/m has to be used as field strength
- Create a Far field recording monitor (Boundary), Section 29.8. 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 between 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**

## 29.25. 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 29.16, 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 
- Switch 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**

## 29.26. S-Parameter calculation

**Usage**

- Create Port(s), Section 29.22
- 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**

## 29.27. 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**



## 29.28. 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](http://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**

## 29.29. 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**

## 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 

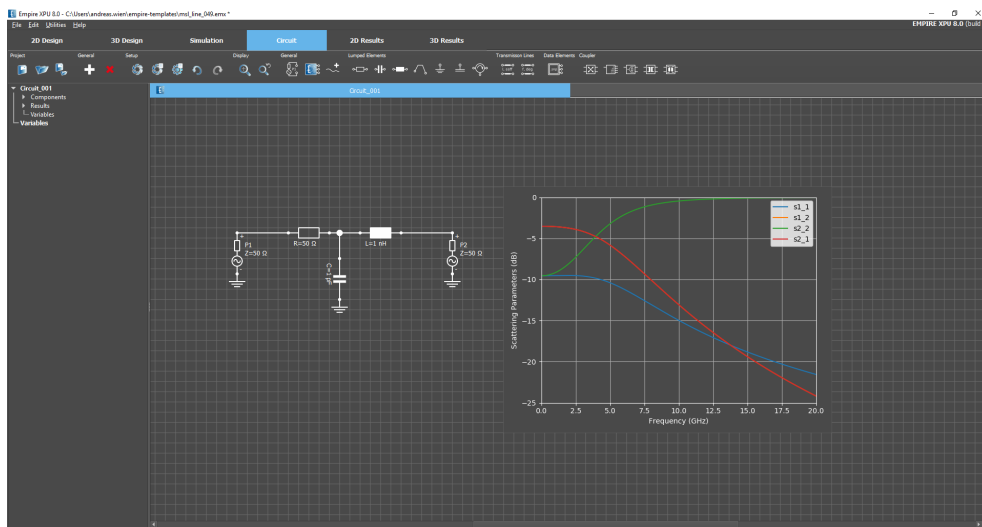






Figure 30.1.: Circuit Simulation Example

- terminations, e.g. ground 
- Ports 
- Click **Run Circuit Simulation** 
- Click **Add Result**  place plot and select S-Parameter or Far Field

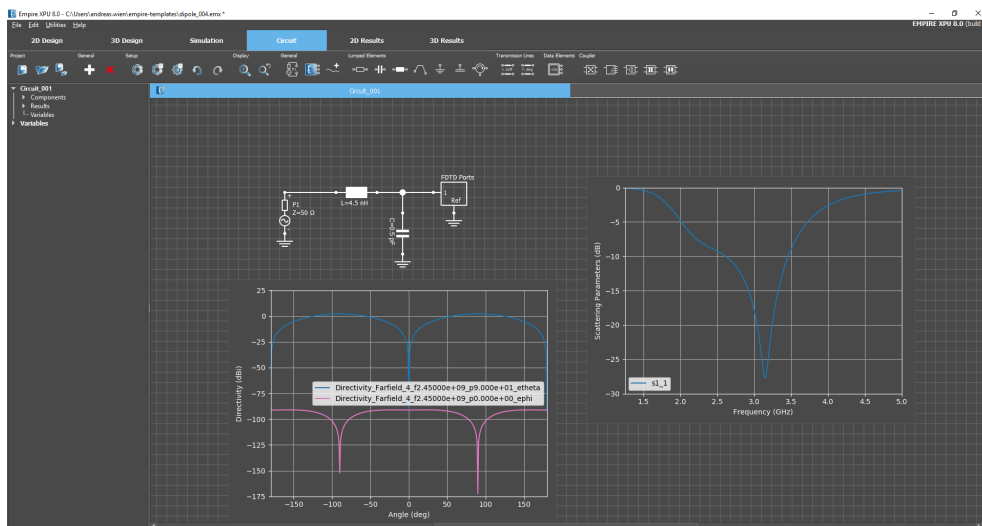



Figure 30.2.: EM-Circuit Co-Simulation Example

See Figure 30.2 for an example (Dipole with matching network)






Figure 30.3.: Circuit Element Library








### 30.3. 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 positions. 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

#### 30.3.1. General



-  **Port** Circuit port to be connected to circuit elements. Port number, impedance and temperature 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

#### 30.3.2. 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

#### 30.3.3. Transmission Lines

Ideal transmission line with line impedance and loss factor to be set in the Parameters

-  **Transmission Line (I, 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.






### 30.3.4. 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.

### 30.3.5. Coupler

Ideal couplers

-  **Phase shift coupler** Impedance, coupling factor and phase shift can be specified
-  **4x90 degree coupler** Impedance can be set.
-  **Wilkinson Divider** Impedance 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.

## 30.4. Circuit Simulation Setup

### 30.4.1. Circuit Simulation Configuration

Click **Circuit Simulation Configuration**  to adjust frequency range and size used for circuit simulation.

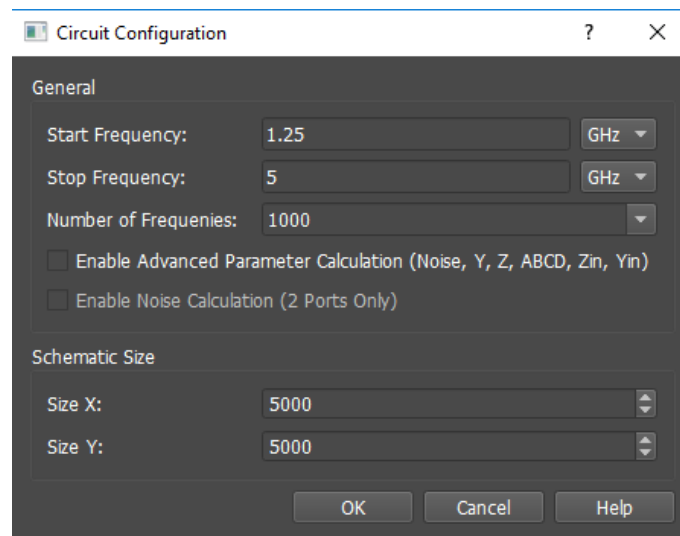


Figure 30.4.: Circuit Simulation Configuration

### 30.4.2. Circuit Editor Options

Click **Circuit Editor Options**  to adjust colors and background for circuit simulation.

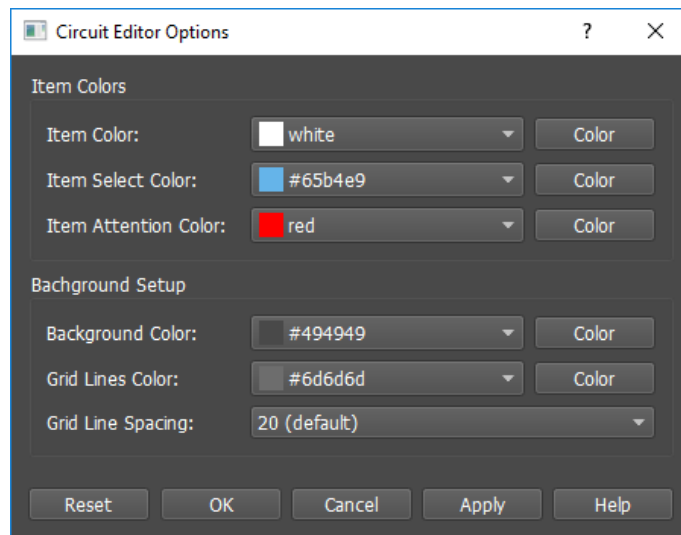



Figure 30.5.: Circuit Editor Options

# 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 section 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 distant 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.

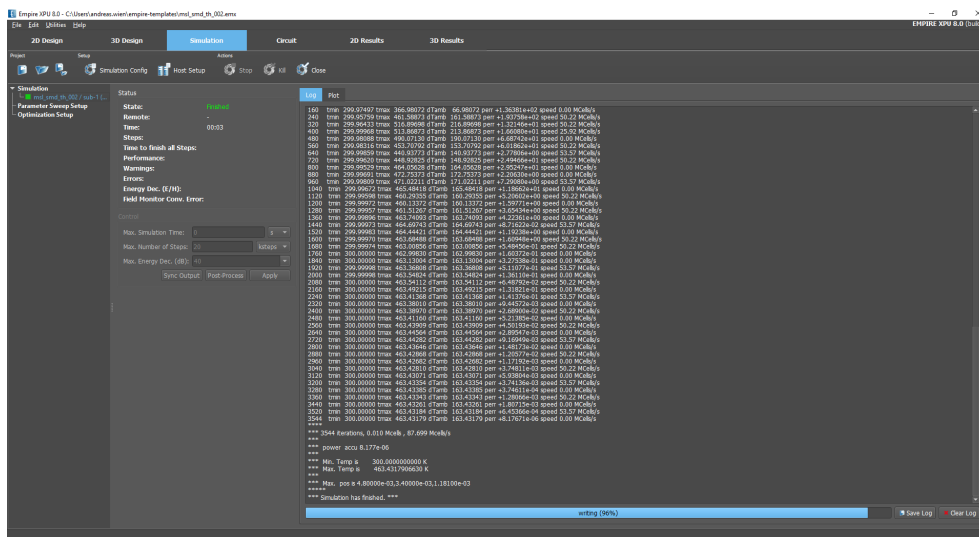


Figure 31.1.: Snapshot of a thermal simulation

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  $x_0$ ,  $y_0$ ,  $z_0$ .

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 ( $x_0=0$ ,  $y_0=0$ ,  $z_0=0$ )

#### Hint

The Origin of the library element ( $x_0, y_0, z_0$ ) 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.

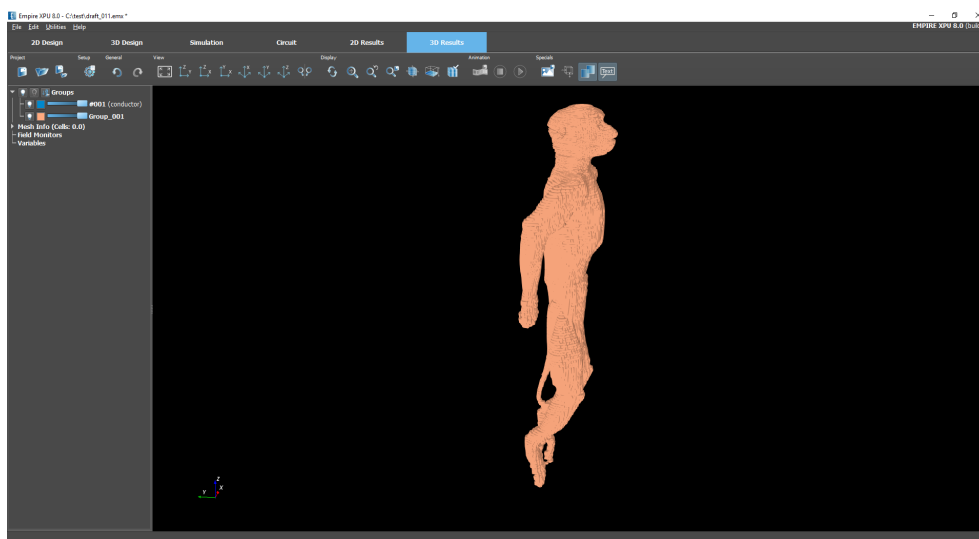


Figure 32.1.: View of the monkey model

**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 objects.

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 structure 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 according 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 axes) 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  $a, a/2, \dots, a/N$  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  $P_n$  between  $P_{n-1}$  and  $P_{n+1}$  is obsolete if it is  
part of the line  $(P_{n-1}, P_{n+1})$

### 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 grid 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 perpendicular 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 minimum 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 Handles 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  
 $0 < \text{phi}0 < 90$  degree

---

**47** Slope Angle phi1=  
Default *50.0*

Anlge at end of bond wire  
 $0 < \text{phi}1 < 90$  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 or subtracted from other objects the circle 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.  $0.2 * \text{delta}$

N = 1 - effective diameter appr.  $1 * \text{delta}$

---

### 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 circle 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 circle 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 separated 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  $0.001 * (\text{max-min})$  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



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 Aspect 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)
  - wavelenghtes 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)
  - wavelenghtes 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)
  - wavelenghtes 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)
  - wavelenghtes 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)
  - wavelenghtes 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)
- wavelenghtes 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     E<sub>tan</sub>=0 at boundary ( dark red line)  
Magnetic    H<sub>tan</sub>=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.  
Touchstone 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.

Separate 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:

$$Q_{\text{perf}} = \rho_{\text{bl}} * C_{\text{bl}} * W_{\text{bl}} * \rho_{\text{tiss}} * (T - T_{\text{bl}})$$

---

**233** Blood Heat Capacity J/kg/K

Default *automatic*

Effects on tissue blood perfusion

Parameter C\_bl used in perfusion rate equation:

$$Q_{\text{perf}} = \rho_{\text{bl}} * C_{\text{bl}} * W_{\text{bl}} * \rho_{\text{tiss}} * (T - T_{\text{bl}})$$

---

**234** Blood Temperature in K

Default *automatic*

Effects on tissue blood perfusion

Parameter T\_bl used in perfusion rate equation:

$$Q_{\text{perf}} = \rho_{\text{bl}} * C_{\text{bl}} * W_{\text{bl}} * \rho_{\text{tiss}} * (T - T_{\text{bl}})$$

---

**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:

Radiation is disregarded

**Part V.**  
**Error Messages**



---

This part gives a summary 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](http://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 available 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 "ieec/C95.3 or ieec/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 Magnetic 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 operation"  
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 permitted 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 permitted 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-neighbor 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 permitted in Bond Wire creation."  
Enter 2D points in the polygon plane for this operation
- 0451 "Only 2D Objects are permitted in CYLINDER creation."  
Enter 2D points in the polygon plane for this operation

- 0452 "Subdivide  $i$  500"  
Decrease the number for subdivision
- 0454 "Symmetry check with 1d Arrows only"  
Only arrows directed in either x, y, or 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, or 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 performed 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  $i$  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 anniation 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: - $z$  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  $i$  0 not allowed"  
Please check the delay definiton of the exciting port
- 3004 "Delay  $i$  1 second found."  
Please check the delay definiton of the exciting port
- 3005 "Diode too short, NRS  $i$ = 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 inhomogenous 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<sup>1</sup>, 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.

$$T_{sim} = \frac{N_{cells}N_{timesteps}}{Performance} \quad (40.1)$$

### 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.

---

<sup>1</sup>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:

$$N_{timesteps} = \frac{T_{pulse} + \frac{D}{c}}{\delta t} \quad (40.2)$$

Here,  $D$  is the diameter of the simulation volume or line length,  $c$  is the velocity of light and  $\delta t$  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

$$N_{cells} = \left(\frac{s \cdot f_{max}}{c}\right)^3 D_x D_y D_z \quad (40.3)$$

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

$$Memory = N_{cells} * 24Bytes \quad (40.4)$$

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.

$$\nabla \times \vec{H} = \dot{\vec{D}} + \vec{J}, \quad (40.5)$$

$$\nabla \times \vec{E} = -\dot{\vec{B}}, \quad (40.6)$$

$$\vec{D} = \epsilon \vec{E}, \quad (40.7)$$

$$\vec{B} = \mu \vec{H}, \quad (40.8)$$

$$\vec{J} = \sigma \vec{E}, \quad (40.9)$$

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  $\varepsilon$ ,  $\mu$ , and  $\sigma$  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  $\dot{B}_x$  component gives

$$\dot{B}_x = \frac{E_y(x, y, z + \frac{\delta z}{2}) - E_y(x, y, z - \frac{\delta z}{2})}{\delta z} - \frac{E_z(x, y + \frac{\delta y}{2}, z) - E_z(x, y - \frac{\delta y}{2}, z)}{\delta y} \quad (40.10)$$

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

$$A(x, y, z, t) = A \left( \sum_i \delta x_i, \sum_j \delta y_j, \sum_k \delta z_k, n\delta t \right) \triangleq A_{i,j,k}^n \quad (40.11)$$

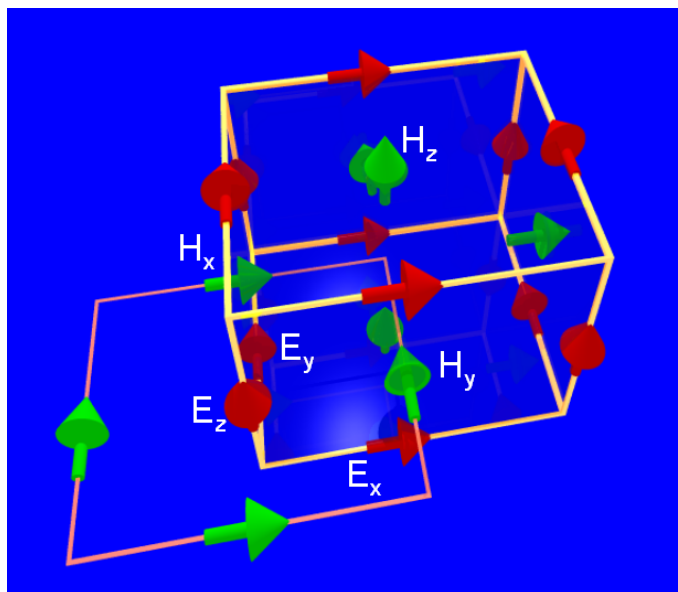


Figure 40.1.: Arrangement of field components in a Yee cell

is introduced. The FDTD scheme can then be written as:

$$E_{x_{i^+,j^+,k}}^{n+1} = \frac{1 - \frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}}{1 + \frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}} E_{x_{i^+,j^+,k}}^n + \frac{2\delta t}{1 + \frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}} \left( \frac{H_{z_{i^+,j^+,k}}^{n+1/2} - H_{z_{i^+,j^-,k}}^{n+1/2}}{\delta y_j + \delta y_{j-1}} - \frac{H_{y_{i^+,j^+,k}}^{n+1/2} - H_{y_{i^+,j^-,k}}^{n+1/2}}{\delta z_k + \delta z_{k-1}} \right) \quad (40.12)$$

$$E_{y_{i^+,j^+,k}}^{n+1} = \frac{1 - \frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}}{1 + \frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}} E_{y_{i^+,j^+,k}}^n + \frac{2\delta t}{1 + \frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}} \left( \frac{H_{x_{i^+,j^+,k}}^{n+1/2} - H_{x_{i^+,j^+,k-}}^{n+1/2}}{\delta z_k + \delta z_{k-1}} - \frac{H_{z_{i^+,j^+,k}}^{n+1/2} - H_{z_{i^-,j^+,k}}^{n+1/2}}{\delta x_i + \delta x_{i-1}} \right) \quad (40.13)$$

$$E_{z_{i^+,j^+,k}}^{n+1} = \frac{1 - \frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}}{1 + \frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}} E_{z_{i^+,j^+,k}}^n + \frac{2\delta t}{1 + \frac{\sigma_{ijk}\delta t}{2\varepsilon_{ijk}}} \left( \frac{H_{y_{i^+,j^+,k}}^{n+1/2} - H_{y_{i^-,j^+,k}}^{n+1/2}}{\delta x_i + \delta x_{i-1}} - \frac{H_{x_{i^+,j^+,k}}^{n+1/2} - H_{x_{i^-,j^-,k}}^{n+1/2}}{\delta y_j + \delta y_{j-1}} \right) \quad (40.14)$$

$$H_{x_{i^+,j^+,k}}^{n+1/2} = H_{x_{i^+,j^+,k}}^{n-1/2} + \frac{\delta t}{\mu_{ijk}} \left( \frac{E_{y_{i^+,j^+,k+1}}^{n+1/2} - E_{y_{i^+,j^+,k}}^{n+1/2}}{\delta z_k} - \frac{E_{z_{i^+,j^+,k+1}}^{n+1/2} - E_{z_{i^+,j^+,k}}^{n+1/2}}{\delta y_j} \right) \quad (40.15)$$

$$H_{y_{i^+,j^+,k}}^{n+1/2} = H_{y_{i^+,j^+,k}}^{n-1/2} + \frac{\delta t}{\mu_{ijk}} \left( \frac{E_{z_{i^+,j^+,k+1}}^{n+1/2} - E_{z_{i^+,j^+,k}}^{n+1/2}}{\delta x_i} - \frac{E_{x_{i^+,j^+,k+1}}^{n+1/2} - E_{x_{i^+,j^+,k}}^{n+1/2}}{\delta z_k} \right) \quad (40.16)$$

$$H_{z_{i^+,j^+,k}}^{n+1/2} = H_{z_{i^+,j^+,k}}^{n-1/2} + \frac{\delta t}{\mu_{ijk}} \left( \frac{E_{x_{i^+,j^+,k}}^{n+1/2} - E_{x_{i^+,j^+,k}}^{n+1/2}}{\delta y_j} - \frac{E_{y_{i^+,j^+,k}}^{n+1/2} - E_{y_{i^+,j^+,k}}^{n+1/2}}{\delta x_i} \right) \quad (40.17)$$

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<sup>1</sup> 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**

---

<sup>1</sup>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.

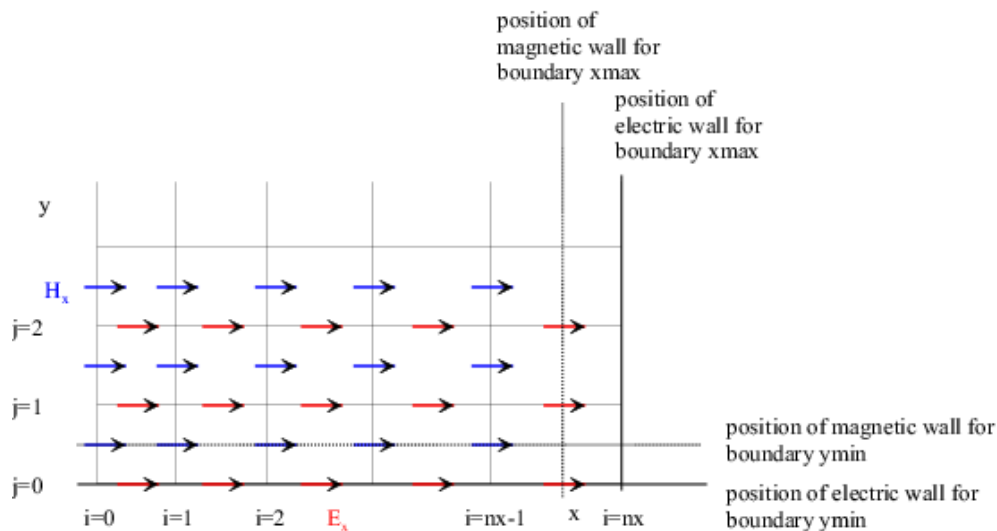


Figure 42.1.: Position of electric and magnetic walls in the grid

**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 **pm1** boundary conditions should be at least a quarter of a wavelength.

**Hint**

The boundary employing **pm1** 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  $\delta t$ .

### 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

$$\delta t \leq \frac{1}{c_0 \sqrt{\frac{1}{(\delta x)^2} + \frac{1}{(\delta y)^2} + \frac{1}{(\delta z)^2}}}. \quad (43.1)$$

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).

$$\delta t \leq \frac{1}{c_0 \max_{i,j,k} \left( \sqrt{\frac{1}{\mu_{xijk} \varepsilon_{xijk} (\delta x_i)^2} + \frac{1}{\mu_{yijk} \varepsilon_{yijk} (\delta y_j)^2} + \frac{1}{\mu_{zijk} \varepsilon_{zijk} (\delta z_k)^2}} \right)} \quad (43.2)$$

### 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.

$$L_{\text{source}} = n_{\text{source}} b_{\text{dis}} \quad (44.1)$$

Here, the parameter  $n_{\text{source}}$  is entered in the parameter setup and  $b_{\text{dis}}$  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<sup>1</sup> when plane waves are used.

---

<sup>1</sup>to simulate a ground plane

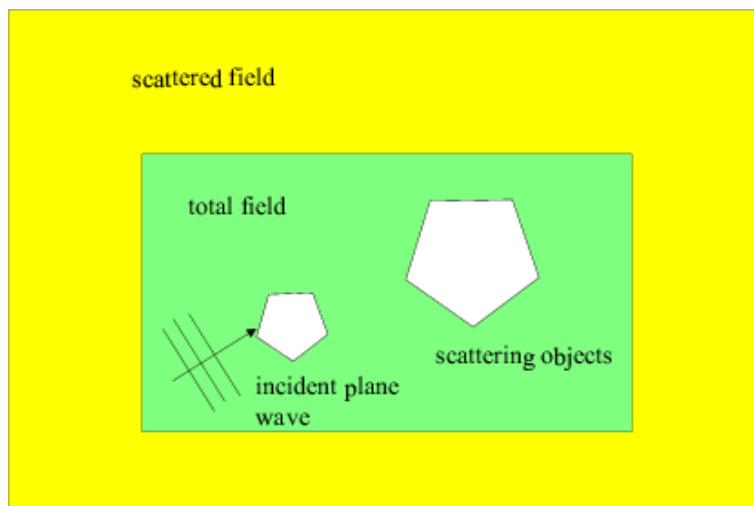


Figure 44.1.: Plane wave excitation

**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  $\theta$  and  $\varphi$  and arbitrary polarization, specified by  $E_\theta$  and  $E_\varphi$ .

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.

$$\begin{pmatrix} E_\theta \\ E_\varphi \end{pmatrix} = \begin{pmatrix} \cos(\varphi) \cos(\theta) & \sin(\varphi) \cos(\theta) & -\sin(\theta) \\ -\sin(\varphi) & \cos(\varphi) & 0 \end{pmatrix} \cdot \begin{pmatrix} E_x \\ E_y \\ E_z \end{pmatrix} \quad (44.2)$$

**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.

$$\beta = \frac{2\pi}{c_0} \sqrt{\varepsilon_r f_0^2 - f_c^2} \quad (44.3)$$

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.

$$\text{exc}(t) = \cos[2\pi f_0(t - t_0)] e^{-\left(\frac{t-t_0}{\tau}\right)^2} \quad (44.4)$$

This definition leads to the following properties of the pulse:

- The amplitudes in frequency domain at the borders  $f_0 \pm f_c$  are  $-20\text{dB}$  of the amplitude of the center frequency  $f_0$ .
- The amplitude in time domain at the first step  $t = 0$  is  $-100\text{dB}$ .
- 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:

$$NS = \text{int}\left(\frac{1}{f_{max} * DT * 2 * N_{over}}\right) \quad (44.5)$$

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.

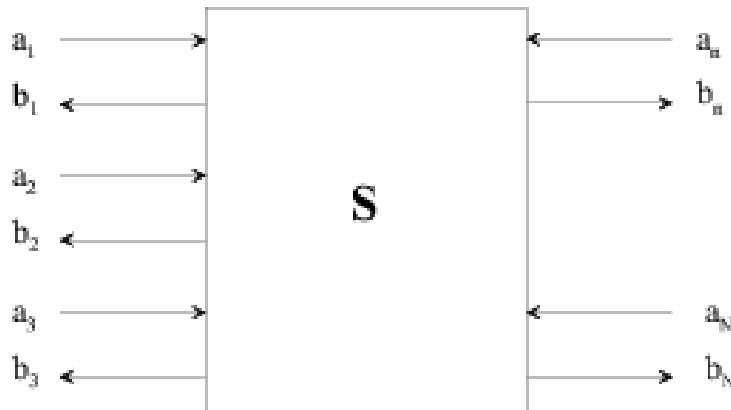


Figure 45.1.: Multiple port scattering matrix

The elements of the scattering matrix are obtained by exciting the structure  $N$  times with the wave quantities  $a_n, n = 1 \dots N$ , under the condition that all ports are terminated by a matched load.

$$s_{i,j} = \left. \frac{b_i}{a_j} \right|_{a_k=0 \forall k \neq j} \quad (45.1)$$

If radiation can occur, as it does for open structures, the leakage of radiated power can be evaluated applying the law of energy conservation.

$$P_{\text{loss}} = \frac{1}{2} \sum_{n=1}^N |a_n|^2 - |b_n|^2 \quad (45.2)$$

### 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<sup>1</sup> where voltages and currents can be uniquely defined. Then the wave quantities can be obtained by

$$a_n = \frac{u_n^{\text{inc}}(f)}{\sqrt{Z_n}} = i_n^{\text{inc}}(f)\sqrt{Z_n} \quad (45.3)$$

$$b_n = \frac{u_n^{\text{ref}}(f)}{\sqrt{Z_n}} = i_n^{\text{ref}}(f)\sqrt{Z_n} \quad (45.4)$$

$$Z_n = \frac{u_n^{\text{inc}}(f)}{i_n^{\text{inc}}(f)} = \frac{u_n^{\text{ref}}(f)}{i_n^{\text{ref}}(f)} \quad (45.5)$$

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

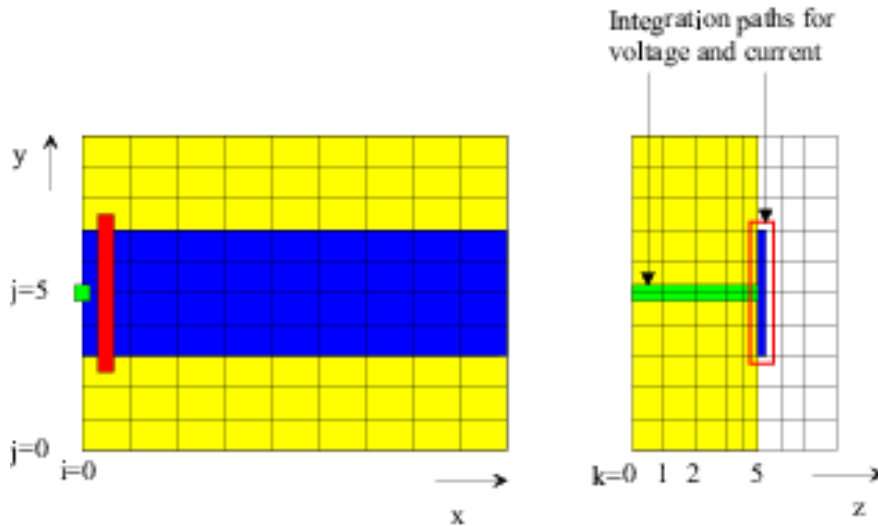


Figure 45.2.: Illustration of an outer microstrip port

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  $j = 5$ .

$$u(t) = \int_0^h E_z dz \approx \sum_{k=0}^5 E_{z0,5,k}^{n\delta t} \delta z_k \quad (45.6)$$

The current is obtained in a similar way by integrating numerically the magnetic field along the conductor,

<sup>1</sup>Hollow waveguides need a special treatment

where the path of integration is specified by the box at  $i = 1/2$  and  $k = 5$ :

$$\begin{aligned}
 i(t) &= \oint_C \vec{H} \cdot d\vec{\ell} \approx \sum_{j=3}^7 H_{y_{0,j,5}}^{(n+1/2)\delta t} \frac{\delta y_j + \delta y_{j-1}}{2} \\
 &\quad - \sum_{j=3}^7 H_{y_{0,j,4}}^{(n+1/2)\delta t} \frac{\delta y_j + \delta y_{j-1}}{2} \\
 &\quad - H_{y_{0,7,5}}^{(n+1/2)\delta t} \frac{\delta z_5 + \delta z_4}{2} \\
 &\quad + H_{y_{0,3,5}}^{(n+1/2)\delta t} \frac{\delta z_5 + \delta z_4}{2}
 \end{aligned} \tag{45.7}$$

### 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.

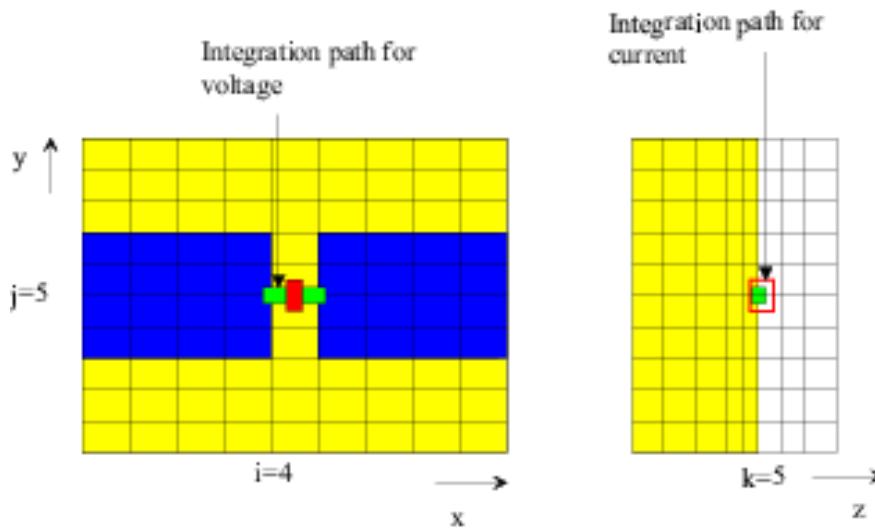


Figure 45.3.: Illustration of an inner port

By default, the characteristic reference impedances are set to  $50\Omega$ .

### 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.

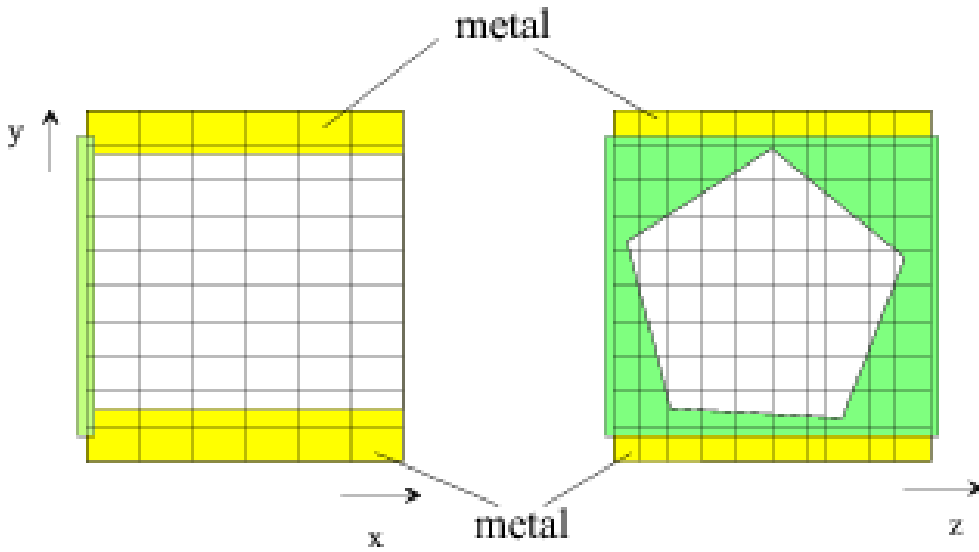


Figure 45.4.: Illustration of a waveguide port

An eigenvalue solver calculates a set of orthogonal eigenvectors which are normalized. For example, modes in the plane  $x_{\min}$  can be expressed as

$$\hat{e}_{n_{j,k}} = \frac{\vec{e}_{n_{j,k}}}{\sqrt{\sum_j \sum_k \vec{e}_{n_{j,k}} \cdot \vec{e}_{n_{j,k}} \delta y_j \delta z_k}} \quad (45.8)$$

$$\hat{h}_{n_{j,k}} = \frac{\vec{h}_{n_{j,k}}}{\sqrt{\sum_j \sum_k \vec{h}_{n_{j,k}} \cdot \vec{h}_{n_{j,k}} \delta y_j \delta z_k}} \quad (45.9)$$

Although voltages and currents can not be uniquely defined for waveguides, it is convenient to record the following expressions:

$$u_n(t) = \sum_j \sum_k \vec{E}_{t_{0,j,k}}^n \cdot \hat{e}_{n_{j,k}} \delta y_j \delta z_k \quad (45.10)$$

$$i_n(t) = \sum_j \sum_k \vec{H}_{t_{0,j,k}}^{n+1/2} \cdot \hat{h}_{n_{j,k}} \delta y_j \delta z_k \quad (45.11)$$

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<sup>2</sup>. Thus, before the simulation run, a proper sampling factor  $n_{\text{sample}}$  will be determined, and only the necessary number of values are stored.

$$u(t) = u(n_{\text{data}} n_{\text{sample}} \delta t) \quad (45.12)$$

where  $n_{\text{data}}$  is the actual data point in the data file ut.

<sup>2</sup>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:

$$ut1.inc \quad ut1 \quad ut2 \quad ut3 \quad (45.13)$$

$$it1.inc \quad it1 \quad it2 \quad it3 \quad (45.14)$$

The reflected waves are simply obtained by

$$ut1.ref = ut1 - ut1.inc \quad (45.15)$$

$$it1.ref = it1 - it1.inc \quad (45.16)$$

### 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:

$$ut1 \quad ut2 \quad ut3 \quad (45.17)$$

$$it1 \quad it2 \quad it3 \quad (45.18)$$

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.

$$ut1.inc = \frac{1}{2}ut1 + \frac{R_{lpi}}{2}it1 \quad (45.19)$$

$$it1.inc = \frac{1}{2}it1 + \frac{1}{2R_{lpi}}ut1 \quad (45.20)$$

$$ut1.ref = ut1 - ut1.inc \quad (45.21)$$

$$it1.ref = it1 - it1.inc \quad (45.22)$$

## 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.

$$u(f) = \sum_{n=0}^{N_{data}} u(t)e^{-2\pi f n m_{sample} \delta t} \delta t \quad (45.23)$$

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:

$$uf1.inc \quad uf1.ref \quad uf1 \quad uf2 \quad uf3 \quad (45.24)$$

$$if1.inc \quad if1.ref \quad if1 \quad if2 \quad if3 \quad (45.25)$$

## 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:

$$Z1 = \frac{uf1.inc}{if1.inc} \quad (45.26)$$

$$Z1.in = \frac{uf1}{if1} \quad (45.27)$$

$$Z2 = \frac{uf2}{if2} \quad (45.28)$$

$$Z3 = \frac{uf3}{if3} \quad (45.29)$$

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:

$$b_1 = \frac{uf1.ref}{\sqrt{Z1}} \quad (45.30)$$

$$b_2 = \frac{uf2}{\sqrt{Z2}} \quad (45.31)$$

$$b_3 = \frac{uf3}{\sqrt{Z3}} \quad (45.32)$$

The incident wave quantities are set to:

$$a_1 = 1 \quad (45.33)$$

$$a_2 = 0 \quad (45.34)$$

$$a_3 = 0 \quad (45.35)$$

## 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:

$$s_{11} = \frac{b_1}{a_1} \quad (45.36)$$

$$s_{21} = \frac{b_2}{a_1} \quad (45.37)$$

$$s_{31} = \frac{b_3}{a_1} \quad (45.38)$$

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  $i = 1, \dots, N$  are excited and each result folder *sub* –  $i$  contains the frequency domain voltages  $\underline{U} = u f_{ij}$  and currents  $\underline{I} = i f_{ij}$  with  $j = 1, \dots, N$ . Then the admittance matrix can be obtained:

$$\underline{Y} = \underline{IU}^{-1} \quad (46.1)$$

With the diagonal characteristic admittance matrix  $Y_{ii}^L = 1/Z_{ii}^L$  and the normalization

$$N_{ij} = \sqrt{\frac{Y_{ii}^L}{Y_{jj}^L}} \quad (46.2)$$

the S-Matrix can be obtained with

$$\underline{S} = \underline{N}(\underline{Y}^L + \underline{Y})^{-1}(\underline{Y}^L - \underline{Y}) \quad (46.3)$$

### 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).

$$\Delta_t H_z + \left(\frac{2\pi f_c}{c_0}\right)^2 H_z = 0 \quad (46.4)$$

$$\Delta_t E_z + \left(\frac{2\pi f_c}{c_0}\right)^2 E_z = 0 \quad (46.5)$$

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).

$$\vec{e}_{n_{i,j}} = \nabla_{\text{t}} H_z \quad (46.6)$$

$$\vec{h}_{n_{i,j}} = \nabla_{\text{t}} E_z \quad (46.7)$$

### 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.

$$\vec{S} = \hat{n} \times \vec{H} \quad (46.8)$$

$$\vec{M} = -\hat{n} \times \vec{E} \quad (46.9)$$

$$N_{\theta} = \iint (S_x \cos \theta \cos \phi + S_y \cos \theta \sin \phi - S_z \sin \theta) e^{jkr' \cos \Psi} ds' \quad (46.10)$$

$$N_{\varphi} = \iint (-S_x \sin \phi + S_y \cos \phi) e^{jkr' \cos \Psi} ds' \quad (46.11)$$

$$L_{\theta} = \iint (M_x \cos \theta \cos \phi + M_y \cos \theta \sin \phi - M_z \sin \theta) e^{jkr' \cos \Psi} ds' \quad (46.12)$$

$$L_{\varphi} = \iint (-M_x \sin \phi + M_y \cos \phi) e^{jkr' \cos \Psi} ds' \quad (46.13)$$

In these equations  $\Psi$  is the angle between observation point and source point.

The far-field is then obtained using (46.14).

$$\bar{E}_{\theta}(\theta, \varphi) = \frac{-k}{\sqrt{8\pi P Z_0}} (L_{\varphi} + Z_0 N_{\theta}) \quad (46.14)$$

$$\bar{E}_{\varphi}(\theta, \varphi) = \frac{k}{\sqrt{8\pi P Z_0}} (L_{\theta} - Z_0 N_{\varphi}) \quad (46.15)$$

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  $\vec{J}$  are determined by the distribution of the electric field  $\vec{E}$ , the mass density  $\rho$  and the electric conductivity  $\sigma$ .

$$\vec{J} = \sigma \vec{E} \quad (46.16)$$

$$SAR = \frac{\sigma |\vec{E}|^2}{\rho} \quad (46.17)$$

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<sup>2</sup> or 100 cm<sup>2</sup>.

### 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 ( $\omega\epsilon \ll \sigma$ ). Considering this prerequisite the simulation is performed with a permittivity of free space ( $\epsilon_r = 1$ ) in order to reduce the simulation time. The propagation speed of an electromagnetic wave inside the human body decreases with increasing  $\epsilon_r$ . 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  $\vec{E}_{Body}$

$$j\omega\epsilon_0 \vec{n} \vec{E}_{Air} = (\sigma + j\omega\epsilon) \vec{n} \vec{E}_{Body} \quad (46.18)$$

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  $\omega$  is determined by the scaled results at an angular frequency of  $\omega'$ .

$$\vec{E}_{Body}(\omega) = \frac{\omega}{\omega'} \frac{\sigma' + j\omega'\epsilon'}{\sigma + j\omega\epsilon} \vec{E}_{Body}(\omega') \quad (46.19)$$

## 46.6. Dielectric Properties of Biological Tissue

In order to consider the frequency-dependent dielectric properties of biological tissue a parametric model from literature<sup>1</sup> has been implemented. The model provides the relative permittivity  $\epsilon_r$  and the electric conductivity  $\sigma$  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.

$$\underline{\epsilon}(\omega) = \epsilon_{\infty} + \sum_{n=1}^4 \frac{\Delta\epsilon_n}{1 + (j\omega\tau_n)^{1-\alpha_n}} + \frac{\sigma}{j\omega\epsilon_0} \quad (46.20)$$

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.

<sup>1</sup>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," 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 ..... 374             |  Create Connectivity Groups ..... 356           |
|  Add Fixed ..... 362                      |  Create Connectivity Groups (discretized) . 356 |
|  Add Fixed Lines ..... 363                |  Create Cylinder (Circular) ..... 333           |
|  Add Port Number(s) ..... 360             |  Create Dimensioning ..... 354                  |
|  Add Property ..... 372                   |  Create Field Monitor ..... 336                 |
|  Add To Discretization ..... 363          |  Create Library Object ..... 334                |
|  Add Unfixed Lines ..... 363              |  Create Line Poly ..... 362                     |
|  Align Object to Arrow ..... 375          |  Create Mesh ..... 343                          |
|  Align Object to Point ..... 375          |  Create Polygon ..... 333                       |
|  Area Measurement ..... 355               |  Create Source ..... 335                        |
|  Arrow 1/3rd Rule ..... 364               |  Cycle Elements ..... 370                       |
|  Arrow Concat (+) ..... 352               |  Delete (Del) ..... 340                         |
|  Arrow Multiple (&) ..... 352             |  Delete Fixed Range ..... 365                   |
|  Arrow Scale (*) ..... 353                |  Delete Last Property ..... 372                 |
|  Arrow Subdivide (/) ..... 353            |  Delete Marks ..... 368                         |
|  Automesh Setup ..... 338                |  Delete Mesh ..... 337                          |
|  Bisect Polygon ..... 369               |  Delete Mesh ..... 365                          |
|  Change Origin ..... 346                |  Delete Mesh Filling ..... 337                 |
|  Change Sign ..... 344                  |  Delete Points ..... 346                      |
|  Change To Poly ..... 345               |  Delete Properties ..... 372                  |
|  Change to Line poly (Script) ..... 361 |  Delete Range ..... 365                       |
|  Check Distance ..... 355               |  Deselect Enclosed (D) ..... 373              |
|  Check Integrity ..... 356              |  Deselect Overlapping (d) ..... 373           |
|  Check Polygon Point Spacing ..... 356  |  Edit Box ..... 354                           |
|  Check Symmetry ..... 356               |  Edit Parameters ..... 340                    |
|  Circle Recognition ..... 370           |  Edit Point/Arrow ..... 342                   |
|  Convert Multibox to Boxes ..... 358    |  Edit Properties ..... 372                    |
|  Convert to Bounding Box ..... 358      |  Edit Substructure ..... 342                  |
|  Convert to Poly ..... 358              |  Edit Text ..... 345                          |
|  Convert to Solid ..... 358             |  Editor Options ..... 339                     |
|  Copy (*) ..... 348                     |  Escape (Esc) ..... 332                       |
|  Copy Marked ..... 366                  |  Explode Library Element ..... 360            |
|  Copy & Mirror (bar) ..... 349          |  Extract Points ..... 344                     |
|  Copy & Rotate ..... 350                |  Extrude Flat Solid ..... 374                 |
|  Copy and Edit Box ..... 354            |  Fix Range ..... 366                          |
|  Copy to Current Group ..... 360        |  Front View ..... 352                         |
|  Create Arrow (-) ..... 353             |  Fullscreen ..... 351                         |
|  Create Bond Wire ..... 362             |  Generic Discretization ..... 367             |
|  Create Box ..... 332                   |  Group Boxes ..... 358                        |
|  Create Circuit Element ..... 336       |  Import ..... 342                             |
|                                                                                                                            |  Insert Point(s) ..... 346                    |
|                                                                                                                            |  Insert Symmetric ..... 343                   |

|                                                                                     |                                      |     |                                                                                     |                                     |     |
|-------------------------------------------------------------------------------------|--------------------------------------|-----|-------------------------------------------------------------------------------------|-------------------------------------|-----|
|    | Intersect (Ctrl y) .....             | 331 |    | Save (Ctrl-s) .....                 | 342 |
|    | Iso X View .....                     | 352 |    | Save View .....                     | 339 |
|    | Iso Y View .....                     | 352 |    | Save & Compile .....                | ??  |
|    | Iso Z View .....                     | 352 |    | Scale to Object Center .....        | 374 |
|    | Last Selection(l) .....              | 347 |    | Scale to the Origin .....           | 374 |
|    | Length Measurement .....             | 357 |    | Select Enclosed (e) .....           | 347 |
|    | Linpoly to Solid .....               | 362 |    | Select Groups' Objects .....        | 360 |
|    | Map Object on Mesh .....             | 357 |    | Select Outside .....                | 373 |
|    | Mark All .....                       | 368 |    | Select Overlapping (p) .....        | 347 |
|    | Mark Range .....                     | 368 |    | Set Height of Current Group .....   | 360 |
|    | Merge .....                          | 331 |    | Set Height(=) .....                 | 351 |
|    | Mesh Info .....                      | 357 |    | Set Parameter by Equation .....     | 341 |
|    | Meshed 3D Structure .....            | 344 |    | Set Parameters by File .....        | 369 |
|    | Mirror (&) .....                     | 349 |    | Set Port Number(s) .....            | 340 |
|    | Move (+) .....                       | 348 |    | Set Property .....                  | 372 |
|    | Move Parameter .....                 | 369 |    | Set Radius .....                    | 371 |
|    | Move to Current Group .....          | 360 |    | Set Tangential .....                | 362 |
|   | Multiple Copy .....                  | 348 |   | Shift Marked .....                  | 366 |
|  | Multiple Subtract .....              | 331 |  | Show Exploded Library Element ..... | 361 |
|  | Negate (m) .....                     | 353 |  | Show Mapped Object .....            | 357 |
|  | Open Current Group .....             | 343 |  | Side View .....                     | 351 |
|  | Open Object (o) .....                | 340 |  | Simplify Polygon .....              | 359 |
|  | Optimize Multibox .....              | 359 |  | Simulation Setup .....              | 338 |
|  | Optimize Solid 1 .....               | 359 |  | Smooth Discretization .....         | 367 |
|  | Optimize Solid 2 .....               | 359 |  | Smooth Discretization Marked .....  | 367 |
|  | Oversize .....                       | 375 |  | Split Solid .....                   | 359 |
|  | Oversize with Roundings .....        | 375 |  | Start Simulation .....              | 338 |
|  | Poly to Rounded Strip .....          | 370 |  | Store Variant .....                 | 338 |
|  | Poly to Strip .....                  | 370 |  | Stretch (/) .....                   | 350 |
|  | Poly to Wire .....                   | 371 |  | Stretch Parameter .....             | 369 |
|  | Propose Discretization .....         | 364 |  | Subdivide .....                     | 344 |
|  | Propose Discretization (local) ..... | 364 |  | Subtract first-second .....         | 331 |
|  | Push Regular Expression .....        | 345 |  | Subtract Port Number(s) .....       | 361 |
|  | Rasterize Polygon .....              | 371 |  | Toggle Mode .....                   | 345 |
|  | Reciprocal .....                     | 345 |  | Top View .....                      | 352 |
|  | Redo .....                           | 341 |  | Undo .....                          | 341 |
|  | Redraw .....                         | 339 |  | Unfix Range .....                   | 366 |
|  | Restore View .....                   | 339 |  | User Vertex Transform .....         | 373 |
|  | Return Center Point .....            | 353 |  | Via Fence .....                     | 355 |
|  | Revolve Polygon around Axis .....    | 346 |  | Zoom Extents (z) .....              | 338 |
|  | Rotate .....                         | 349 |  | Zoom Window (z) .....               | 339 |
|  | Rotate Structure and Mesh .....      | 375 |                                                                                     |                                     |     |

## List of Options

|                                        |     |                                                     |     |
|----------------------------------------|-----|-----------------------------------------------------|-----|
| 2D Draft Display .....                 | 388 | GDSII Scale .....                                   | 394 |
| 3D Display Linewidth .....             | 385 | GERBER Aperture Size (mm) .....                     | 395 |
| 3D Measure Cone size .....             | 393 | GERBER Float Format .....                           | 395 |
| 3D Measure Distance .....              | 393 | GERBER Merge Tolerance .....                        | 400 |
| 3D Measure Font size .....             | 393 | GERBER Scale .....                                  | 394 |
| 3D Measure Line Width .....            | 393 | GYM Scale Factor .....                              | 401 |
| Accept Zero Area Objects .....         | 399 | GYM Shift (x y z) .....                             | 401 |
| Arc Recognition min. # Segments .....  | 394 | Galvo speed .....                                   | 397 |
| Arc Resolution (DEG) .....             | 398 | Group Drawing Order .....                           | 380 |
| Arrow Distance .....                   | 392 | HPGL pen width .....                                | 401 |
| Arrow Heads .....                      | 392 | HPGL scale .....                                    | 396 |
| Arrow Rectangle Color .....            | 377 | Horizontal Component Snap .....                     | 377 |
| Arrow Text .....                       | 377 | Horizontal Mesh Snap .....                          | 379 |
| Arrow color .....                      | 377 | Horizontal Snap .....                               | 379 |
| Automatic Circles .....                | 386 | Icon Size .....                                     | 402 |
| Background Color .....                 | 380 | Ignore layer in DXF block .....                     | 399 |
| Background Color .....                 | 384 | Import Scale Factor .....                           | 398 |
| Background Gradient .....              | 384 | Lamp Current (A) .....                              | 398 |
| Circle Recognition Points .....        | 399 | Laser off speed .....                               | 397 |
| Circle Recognition Tolerance .....     | 399 | Laser on speed .....                                | 397 |
| Circle Recognition min. # Points ..... | 387 | Light 0 Ambient .....                               | 382 |
| Circle Recognition min. # Points ..... | 394 | Light 0 Diffuse .....                               | 383 |
| Circle Resolution (Degrees) .....      | 386 | Light 0 Position .....                              | 382 |
| Circle Tolerance .....                 | 394 | Light 0 Specular .....                              | 383 |
| Circle diameter Scale .....            | 397 | Light 1 Ambient .....                               | 383 |
| Circle min # points .....              | 396 | Light 1 Diffuse .....                               | 383 |
| Circle min # points .....              | 397 | Light 1 Position .....                              | 383 |
| Circle tolerance .....                 | 396 | Light 1 Specular .....                              | 383 |
| Circle tolerance .....                 | 397 | Light Global Ambient .....                          | 383 |
| Coordinate Shift .....                 | 398 | Line Poly Default Width .....                       | 386 |
| Create Backup Files .....              | 402 | Linewidth .....                                     | 382 |
| Cursor Color .....                     | 378 | Logging .....                                       | 391 |
| Cursor Size .....                      | 378 | Max Circle diameter .....                           | 397 |
| DRC Failure Behavior .....             | 391 | Max Number of Solid Handles .....                   | 388 |
| DXF Connection Tolerance .....         | 399 | Max instructions per file .....                     | 397 |
| DXF General Tolerance .....            | 399 | Max. Number of Handles .....                        | 382 |
| DXF Scale .....                        | 394 | Max. Number of curves with full table display ..... | 381 |
| Design Check Tolerance .....           | 391 | Maximum Iterations .....                            | 390 |
| Dimensioning Number of Digits .....    | 392 | Merge GERBER Layers .....                           | 400 |
| Dimensioning Scale Factor .....        | 392 | Min. Normal Angle for Wireframe .....               | 381 |
| Dimensioning Tic Size .....            | 392 | Min. Normal Angle in 2D display .....               | 387 |
| Display Grid .....                     | 381 | Minimum Auto Thickness .....                        | 391 |
| Display Mesh .....                     | 381 | Multiple Copy Expression .....                      | 392 |
| Display Mode .....                     | 384 | Number of Digits .....                              | 378 |
| Distance to Poly .....                 | 389 | Number of Points in Bond Wire .....                 | 385 |
| Double Via Holes .....                 | 397 | Output Format .....                                 | 396 |
| EXCELLON decimal digits .....          | 401 | POSTSCRIPT Page Width .....                         | 398 |
| Font Size .....                        | 393 | PS Layout Scale .....                               | 398 |
| GDS VIA Distance (GDS units) .....     | 400 | PS Layout shift .....                               | 398 |
| GDS VIA Size (GDS units) .....         | 400 | Pad Group(s) .....                                  | 390 |
| GDS min. Obj size (GDS units) .....    | 400 | Pad Width .....                                     | 389 |
|                                        |     | Point Text .....                                    | 380 |
|                                        |     | Point color .....                                   | 380 |
|                                        |     | Point size .....                                    | 380 |



|                                     |     |
|-------------------------------------|-----|
| Pointsize .....                     | 382 |
| Poly Minimum Angle .....            | 385 |
| Poly Rasterize .....                | 385 |
| Poly Set Radius Option .....        | 386 |
| Poly Tolerance .....                | 397 |
| Poly tolerance .....                | 387 |
| Pulse Frequency (kHz) .....         | 398 |
| Pulse Length (s) .....              | 398 |
| Relative Accuracy .....             | 388 |
| Repetitions .....                   | 396 |
| Resolution .....                    | 384 |
| Resolution Factor .....             | 393 |
| Resolution Factor .....             | 395 |
| Rotation Angle .....                | 391 |
| Rotpoly Fast Discretization .....   | 387 |
| Rotpoly Subdivision (Integer) ..... | 386 |
| STL Scale .....                     | 395 |
| STL Shift (x y z) .....             | 400 |
| STL Simplification .....            | 401 |
| Save all Triangles in ASCII .....   | 388 |
| Scale Factor .....                  | 396 |
| Scaling Factor .....                | 396 |
| Separate STL Files .....            | 395 |
| Separate STL Objects .....          | 400 |
| Show Bounding Box .....             | 387 |
| Show Coordinate System .....        | 395 |
| Show Mesh Coordinate .....          | 378 |
| Slope Angle phi0= .....             | 385 |
| Slope Angle phi1= .....             | 385 |
| Snap Mode .....                     | 377 |
| Snap on distant Components .....    | 378 |
| Snapped Cursor Color .....          | 379 |
| Software 3D antialiasing .....      | 384 |
| Speed in ms .....                   | 382 |
| Start Coordinates .....             | 396 |
| Start Coordinates .....             | 396 |
| Subroutine Optimization .....       | 397 |
| Transparency Setup .....            | 384 |
| Undersize .....                     | 397 |
| Unit .....                          | 396 |
| Use Attractors .....                | 390 |
| Use slow DRC check .....            | 390 |
| Vertical Component Snap .....       | 378 |
| Vertical Mesh Snap .....            | 379 |
| Vertical Snap .....                 | 379 |
| Via A Group(s) .....                | 389 |
| Via B Group(s) .....                | 390 |
| Via Diameter .....                  | 389 |
| Via Distance .....                  | 389 |
| Via # Points .....                  | 390 |
| Warn for Improper Objects .....     | 388 |
| Wheel Zoom Factor .....             | 380 |

## List of Autodisc Options

|                                    |     |
|------------------------------------|-----|
| Absolute min in units .....        | 405 |
| Arc Resolution in Degrees .....    | 405 |
| Edge accuracy (Percent) .....      | 407 |
| Flat Metal Edge correction .....   | 403 |
| Flat metal Recognition Limit ..... | 407 |
| Max Resolution in units .....      | 405 |
| Max. scan points per Object .....  | 407 |
| Mesh Resolution (Numeric) .....    | 403 |
| Min Cells per Wavelength .....     | 404 |
| More Lines between Edges .....     | 403 |
| Object Roughness (cells) .....     | 404 |
| Object Scan Angle (Degrees) .....  | 403 |
| Object cells 1 .....               | 406 |
| Object cells 2 .....               | 406 |
| Object cells 3 .....               | 406 |
| Object cells 4 .....               | 406 |
| Planar Direction .....             | 405 |
| Refinement ratio .....             | 406 |
| Relative Drawing Accuracy .....    | 404 |
| Relative Limit for Warning .....   | 404 |
| Resolve Curved Geometry .....      | 403 |
| Target min in units .....          | 405 |
| User Defined Simulation Box .....  | 409 |
| Wedge limit in Degrees .....       | 407 |
| max .....                          | 409 |
| max .....                          | 409 |
| max. Aspect ratio .....            | 407 |
| min .....                          | 409 |
| min .....                          | 409 |
| x mesh in units .....              | 407 |
| x mesh options in units .....      | 408 |
| y mesh in units .....              | 408 |
| y mesh options in units .....      | 408 |
| z mesh in units .....              | 408 |
| z mesh options in units .....      | 408 |
| zmax .....                         | 410 |
| zmin .....                         | 409 |

## List of Simulation Options

|                                  |     |
|----------------------------------|-----|
| AC only simulation .....         | 411 |
| Accuracy .....                   | 419 |
| Algorithm .....                  | 418 |
| Ambient Temperature .....        | 419 |
| Blood Density kg/m**3 .....      | 420 |
| Blood Heat Capacity J/kg/K ..... | 420 |
| Blood Temperature in K .....     | 420 |
| Conductors .....                 | 417 |
| Convection .....                 | 420 |

---

|                                    |     |
|------------------------------------|-----|
| Dielectrics .....                  | 417 |
| Drawing Unit .....                 | 416 |
| End Frequency .....                | 415 |
| Energy Decay .....                 | 411 |
| Field Convergence Check .....      | 411 |
| Frequencies .....                  | 413 |
| Maximum Temperature .....          | 419 |
| Mesh Resolution .....              | 416 |
| Mode .....                         | 418 |
| Number for Frequency Points .....  | 415 |
| PGA .....                          | 411 |
| Permittivity .....                 | 413 |
| Radiation .....                    | 421 |
| Resolve Metal sheets .....         | 418 |
| Sides on Ambient Temperature ..... | 420 |
| Simulation Mode .....              | 419 |
| Solvers .....                      | 416 |
| Start Frequency .....              | 415 |
| Structure Type .....               | 416 |
| Target Frequency .....             | 415 |
| Temperature Storage .....          | 419 |
| xmax .....                         | 412 |
| xmax .....                         | 414 |
| xmin .....                         | 412 |
| xmin .....                         | 413 |
| ymax .....                         | 412 |
| ymax .....                         | 414 |
| ymin .....                         | 412 |
| ymin .....                         | 414 |
| zmax .....                         | 413 |
| zmax .....                         | 414 |
| zmin .....                         | 412 |
| zmin .....                         | 414 |