EM-TWIN
Getting Started

Version 9.1.0

September 5, 2024

Contents

1 Introduction

EM-TWIN™ is a versatile 3D electromagnetic field simulator based on the powerful 3D Finite Difference Time Domain Method (FDTD). It is intended as a rapid setup of a simulation using EM sources based on equivalent currents acting as digital twins of measured or simulated antennas[1].

EM-TWIN™ features all state-of-the-art field solver operations, like 2D/3D data import and export, easy structure set-up, automatic meshing, parameterizations, batch processing, remote control and optimization, which makes a rapid analysis of a Digital Twin possible.

The EM-TWIN™ software

- is a tool developed by IMST specialized for Antenna Digital Twin EM simulation
- is based on IMST's FDTD solver EMPIRE XPU™ with its unique innovative software acceleration XPU technology
- applies special patented algorithms (patent by IMST and Rohde&Schwarz) for accurate simulation using EM sources based on equivalent currents
- has special templates for Antenna Digital Twin simulation setup and postprocessing to ease the usage (e.g. Digital Twin for Automotive)
- is available as a stand alone product and as an add-on to EMPIRE XPU™ and compatible to Paiyue™
- supports seamless import of Rohde&Schwarz AMS32 data generated during measurements of antennas in Rohde&Schwarz chambers

Due to its unique on-the-fly compilation and multiple time-stepping approach EM-TWIN™ is able to extensively exploit modern processor architectures which leads to a performance up to 90000 million cells per second (MCells/s)[2]. This performance enables the analysis of large-scale problems such as

- Near- and Farfield simulation for automotive and aerospace

[1] A digital twin is a digital model of an intended or actual real-world physical product, system, or process (a physical twin) that serves as the effectively indistinguishable digital counterpart of it for practical purposes, such as simulation, integration, testing, monitoring, and maintenance.

[2] measured on an AMD Dual Epyc 9684x workstation

- Exposure evaluation of SAR and Absorbed Power Density for mmWave

using EM sources as antenna Digital Twins in shortest simulation time.

1.1 About this document

This booklet is intended to guide the user through the first steps with EM-TWIN™.

The philosophy and the major features of the program will be explained, added with hints, references to the manual and further documents.

2 Installation

2.1 Software Installation

You must have Administration rights to allow all files to be installed.

1. On a Windows system double-click **setup.exe** in the media[1]. The following screen will appear

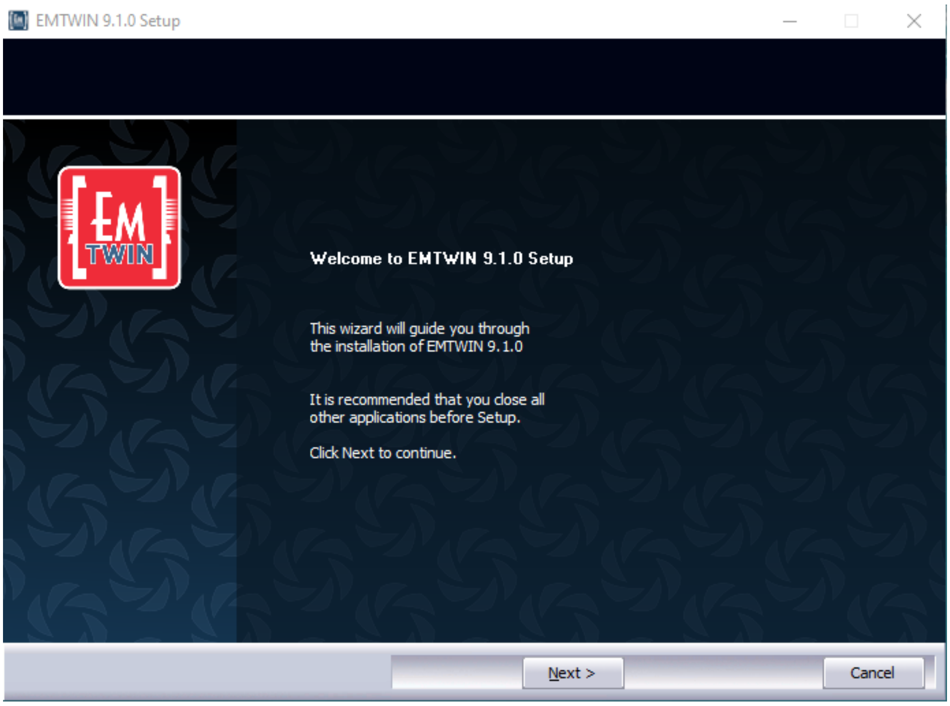


Figure 2.1: Installation welcome screen

[1]Please close running Python programs or Flexlm services

2. Press `Next>` to continue the installation
3. Press `I Agree` to accept license conditions to proceed.
4. Choose Components to be installed. We recommend to perform the full EM-TWIN™ installation. If you do not want to install the Examples or 3D CAD converter, deselect the correspondent check box.
5. Press `Next>` to continue the installation. The following window will appear on your screen:

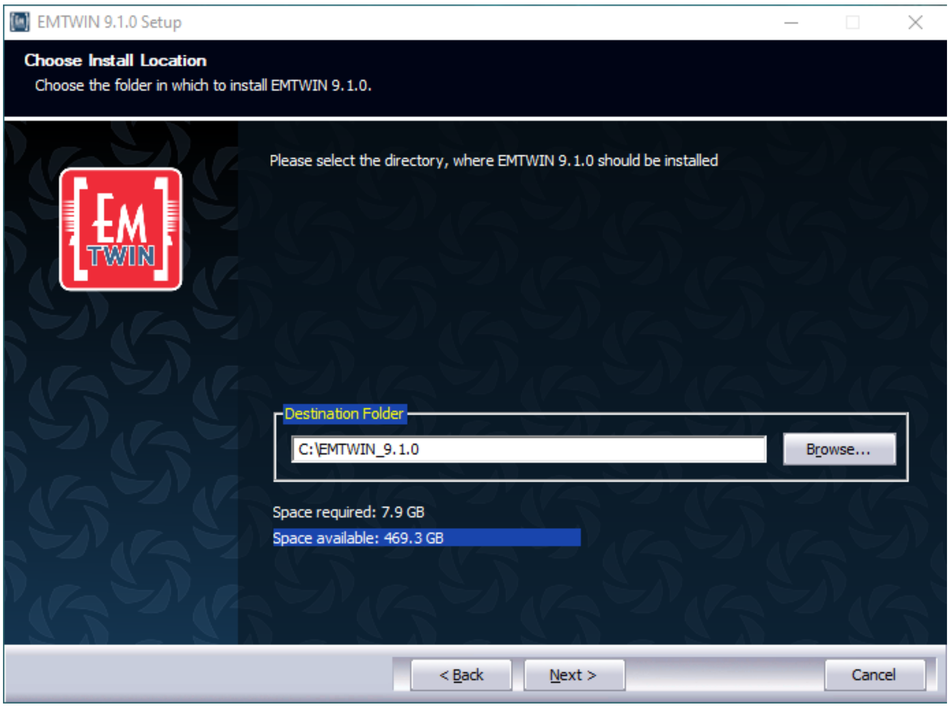


Figure 2.2: Installation destination folder

6. Please choose the **Installation Folder**[2] and check the disk space requirements. Press `Next>` to continue the installation.
7. Finally press `Install`, `Next>`, and `Finish` to exit from the setup window.

2.2 First License Installation (node-locked)

Without a license, the program can only be started for browsing existing projects. Saving and running projects require a valid license. To generate the license key for

[2]Note, on some systems spaces or special characters are not allowed in the installation path

EM-TWIN™ software, some identification numbers from your computer (on which the software shall be installed) are required.

1. Double-click the EM-TWIN™ desktop icon. The following start-up window will appear

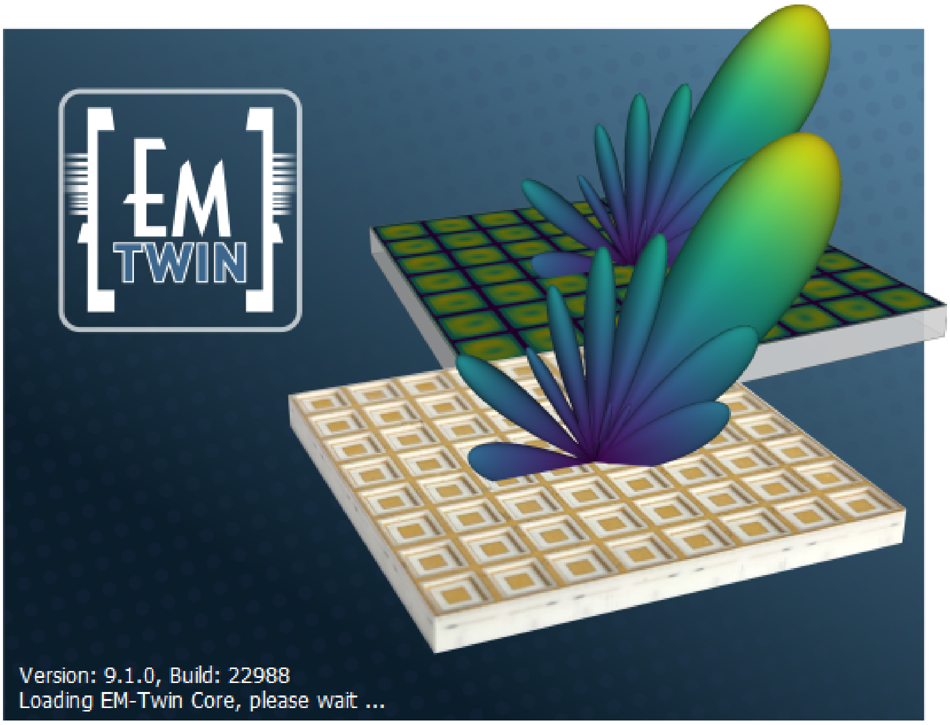


Figure 2.3: Startup window

If no license can be found or a license has expired the following window will come up

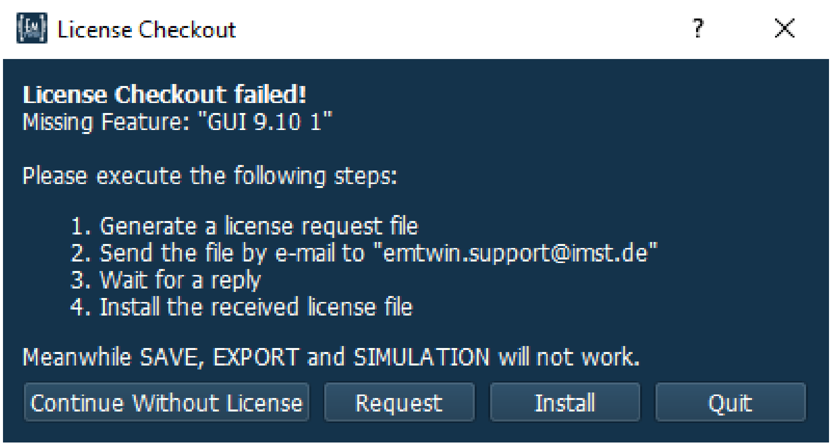


Figure 2.4: License check window

2. Press `Request`, the license request form will open as

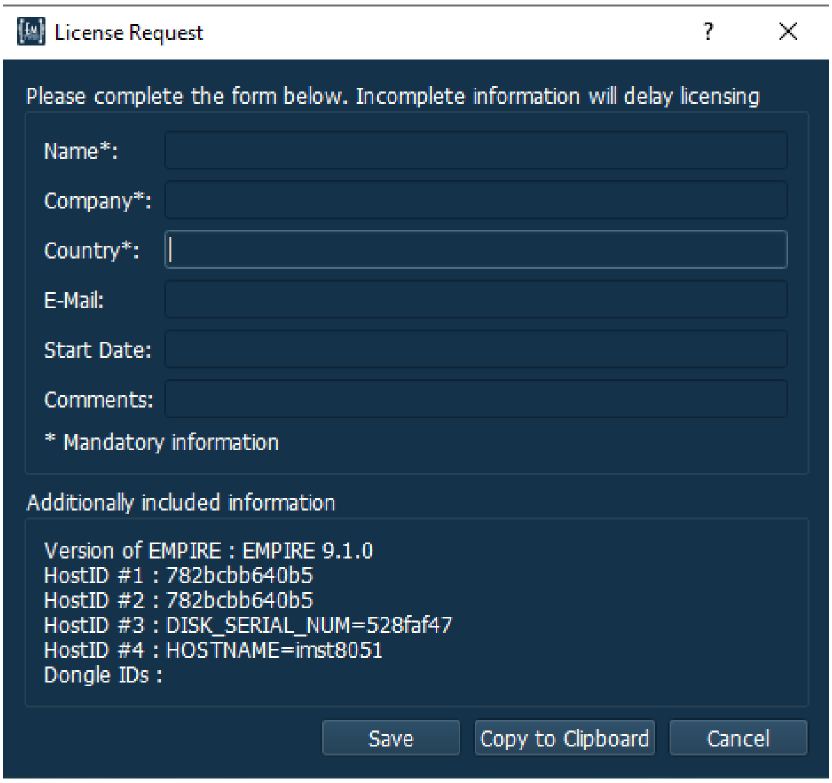


Figure 2.5: License request form

3. Please complete the form and click `Save`. Store the file and send it to `emtwin.support@imst.de`

   The EM-TWIN™ Support Team will generate a license file and send it to you. In order to install the license file:

4. Save the license file that you received by e-mail.
5. Double-click the EM-TWIN™ desktop icon.
6. Select `Install`.
7. Accept the license agreement and select the license file and confirm dialog boxes.
8. The license installation is completed. Close EM-TWIN™ and start it again.

Hint

More information about installation and licensing is available in **EMPIRE-Manual.pdf - Part I: Software Administration**

3 User Interface

When starting an EM-TWIN™ instance from the desktop icon, the Start window will be displayed.

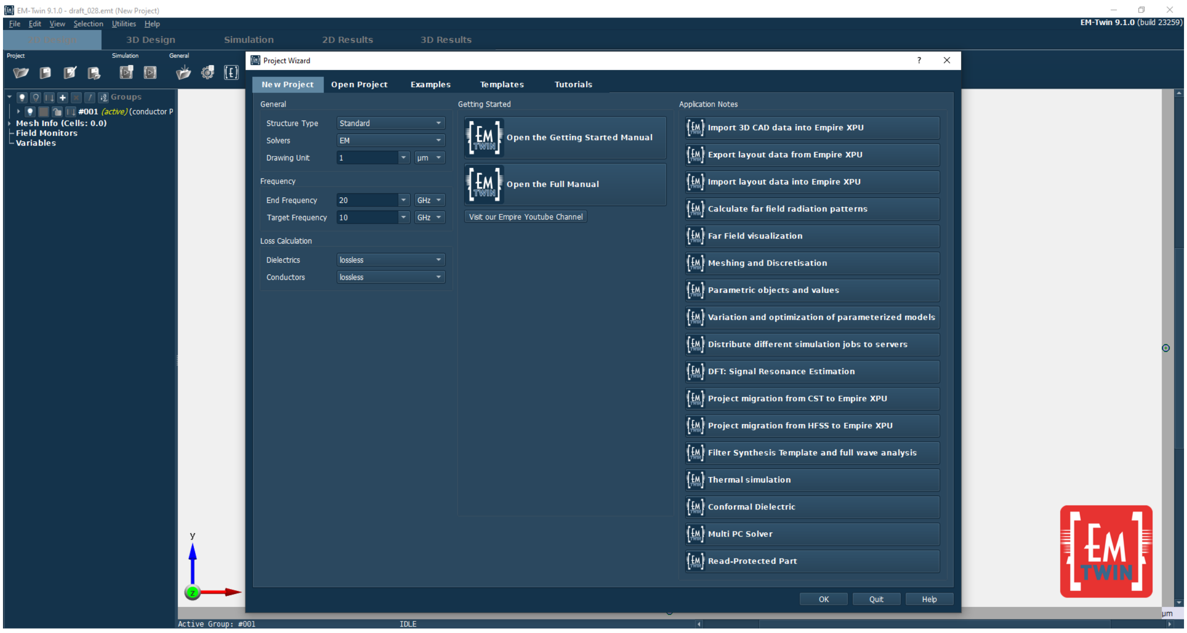


Figure 3.1: Project wizard

The start window allows the user to:

- Create a new project
- Open an existing project
- Load one of the built-in templates, examples or step-by-step tutorials

In the start window, some general parameters for a new project can be set (drawing unit, frequencies, loss treatment, etc.). The end frequency determines the maximum cell spacing. The target frequency is used as pulse center frequency and as default for field monitor storage. During the simulation, voltages and currents are recorded, which yield the frequency parameters in the post processing. Further, near fields are recorded for the target frequency, which can be post processed to obtain far field patterns or SAR (Specific Absorption Rate) and sPD (spatial averaged Power Density) distributions.

When clicking the OK button in tab **New Project** the following window will be displayed.

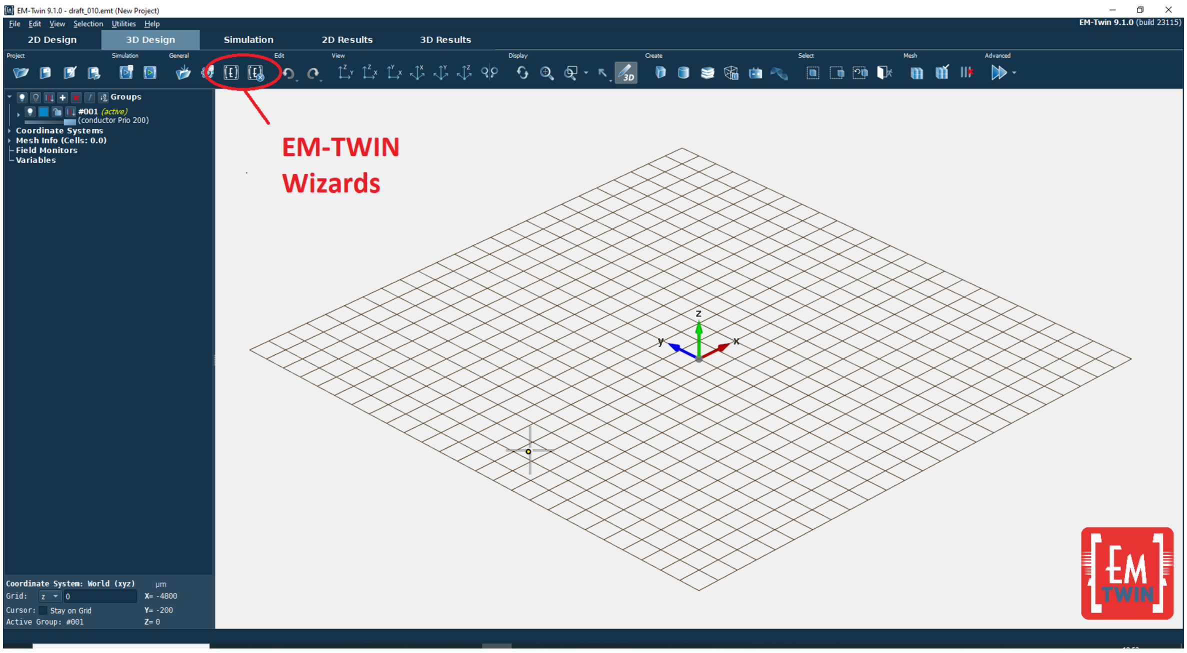


Figure 3.2: New project window

It is recommended to save a new project into a new folder.

3.1 EM-TWIN™ Wizards

The EM-TWIN™ user interface is optimized to set up a digital twin using advanced wizards for

- (Automotive) CAD import and placing an antenna digital twin field source on the CAD surface
- Exposure investigations, where incident or absorbed power density or SAR can be determined using an antenna digital twin field source

The wizards are intended to guide the user through the modelling workflow with very few steps.

3.1.1 EM-TWIN™ Wizard

This wizard requires basically 4 steps to setup an EM-TWIN™ simulation for a near field source placed on a CAD model.

1. Import CAD model
2. Setup material of the CAD model
3. Setup near field source
4. Setup simulation parameters

Step 1

By clicking the EM-TWIN™ Wizard icon the following window comes up.

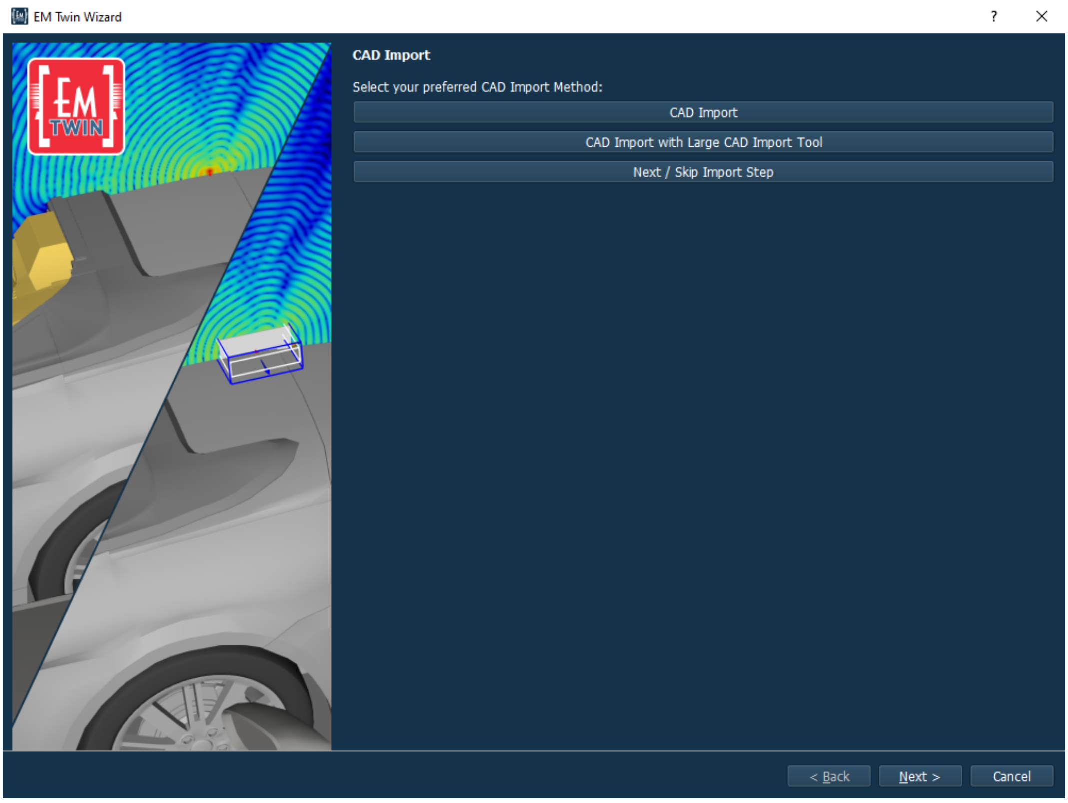


Figure 3.3: EM-TWIN wizard

At first step the CAD Import can be run with a preferred import method.

- **CAD Import**
  Import CAD files without simplification, see Chapter 4.1 for details.
- **CAD Import with Large CAD Import Tool**
  Import large CAD files with the option to reduce complexity, see Chapter 4.1.4 for details.
- **Next / Skip Import Step**
  Continue without CAD import.

After this step, the imported geometry is displayed in the background window.

Step 2

As a second step the available groups are displayed and can be assigned to their respective material properties. After import all objects are assigned as PEC conductors by default. (If objects are arranged into subgroups they inherit the property from their parent group). So only non-PEC groups have to be changed as indicated in Figure 3.4.

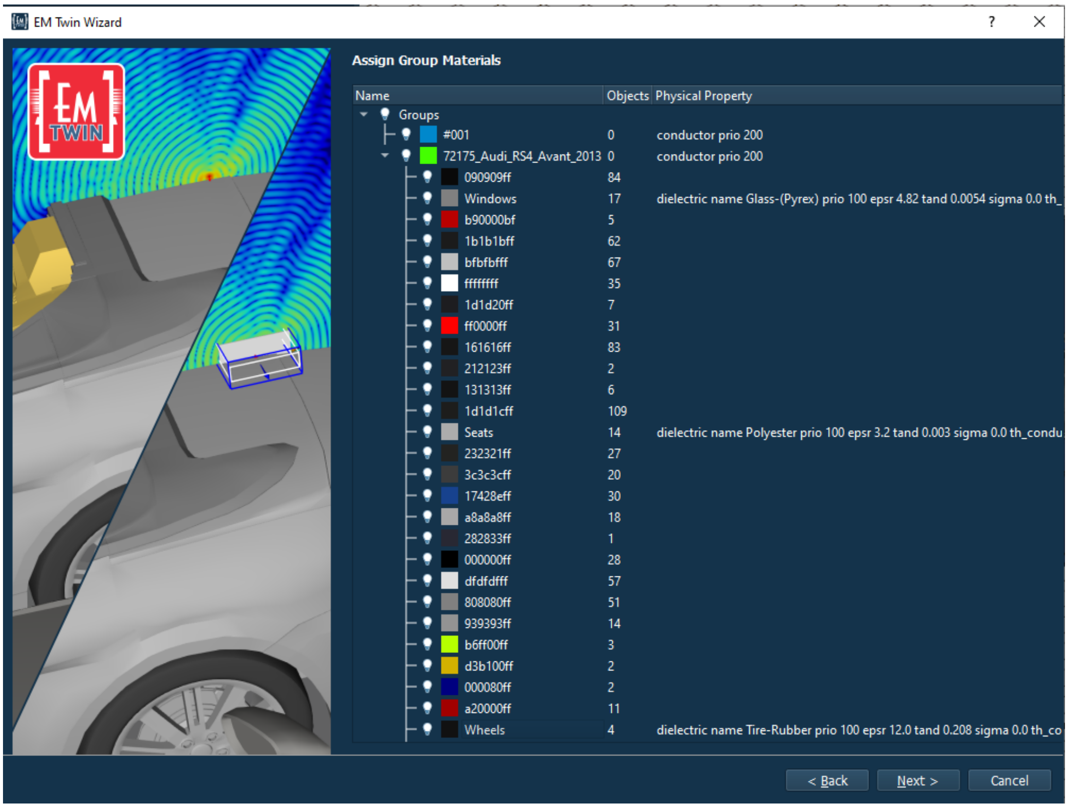


Figure 3.4: Group materials

- **Name**
  Name and color of the group as defined in the CAD file. The group name can be changed with a double-click. The visibility can be controlled with the light bulb.
- **Objects**
  Number of objects available in the group
- **Physical Property**
  Electric parameters (conductor or dielectric) and geometric priority for all objects in this group. The property can be changed with a double-click. With a right click properties can be copied and pasted to other groups.

The group structure may be rearranged by drag&drop groups. New groups can be added with a right click. By hovering the pointer over the groups the respective objects are highlighted in the background window which is useful for identification and setting the electric properties.

All these actions can also be done after completing the wizard in the group tree, see Chapter 4.2 for details.

Step 3

As a third step the antenna field source can be selected, configured and placed at the desired location. The field source can be recognized by a file with ending **.surf.dat**. If not already available this file can be created from near field data files.

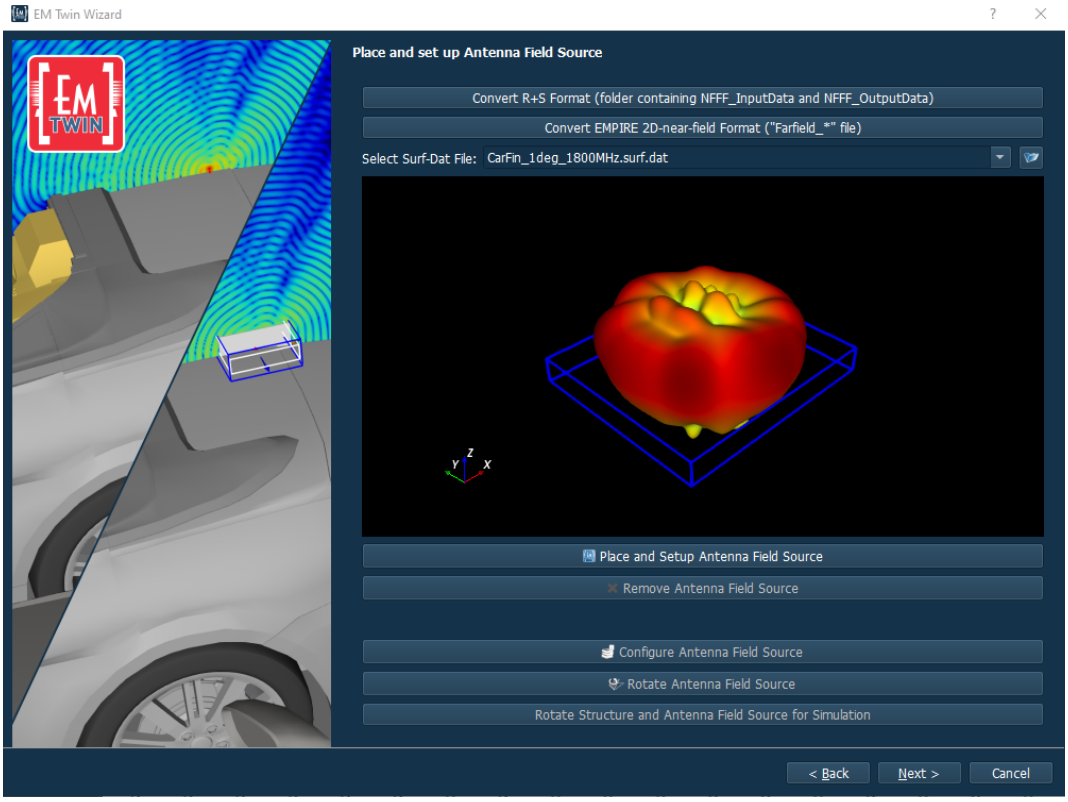


Figure 3.5: Antenna Field Source setup wizard

- **Convert R+S Format**
  Select a folder **x** containing near field data from Rohde&Schwarz and create a **Surf-Dat** source file.
  The following data files are required either altogether in a dedicated folder or in the following subfolder structure:
  - Mesh files .nas in **x**\NFFF_InputData
  - Configuration file .hyb in **x**\NFFF_OutputData*Frequency*
  - Electric Surface Currents .hmj in **x**\NFFF_OutputData*Frequency*
  - Magnetic Surface Currents .hmm in **x**\NFFF_OutputData*Frequency*
  - Optional 3D radiation pattern .cut in **x**\NFFF_OutputData*Frequency*
- **Convert EMPIRE 2D-nearfield Format**
  Select a boundary near field file (Farfield_) from an EMPIRE simulation and create a **Surf-Dat** source file.
- **Select Surf-Dat File**
  Load a Surf-Dat file obtained by conversion or browse for an existing file.

- **Place and Setup Antenna Field Source**
  After clicking, pick a point in the structure which represents the bottom center of the field source (typically antenna location). A local coordinate system **uvw** will be used which represents the orientation of the field source box with the vector $\vec{w}$ normal to the picked surface. All view options, like group visibility, zoom, pan and rotation are supported for point selection.
- **Remove Antenna Field Source**
  Useful for repositioning of entered field source.
- **Configure Antenna Field Source**
  Specify port number, and optional reflector size and power measurement oversize values. If the source is placed on a metal surface, the respective side of the source should be disabled (typically zmin). For more details see Chapter 4.3.
- **Rotate Antenna Field Source**
  Rotate field source around $\vec{u}$, $\vec{v}$ or $\vec{w}$ to align radiation direction with respect to CAD model
- **Rotate Structure and Antenna Field Source for Simulation**
  If the field source box is not parallel to the world coordinate system, it has to be rotated including the completre structure. Using this button automatically rotates the source and CAD model so the field source is properly aligned.

Step 4

As the last step simulation parameters have to be set

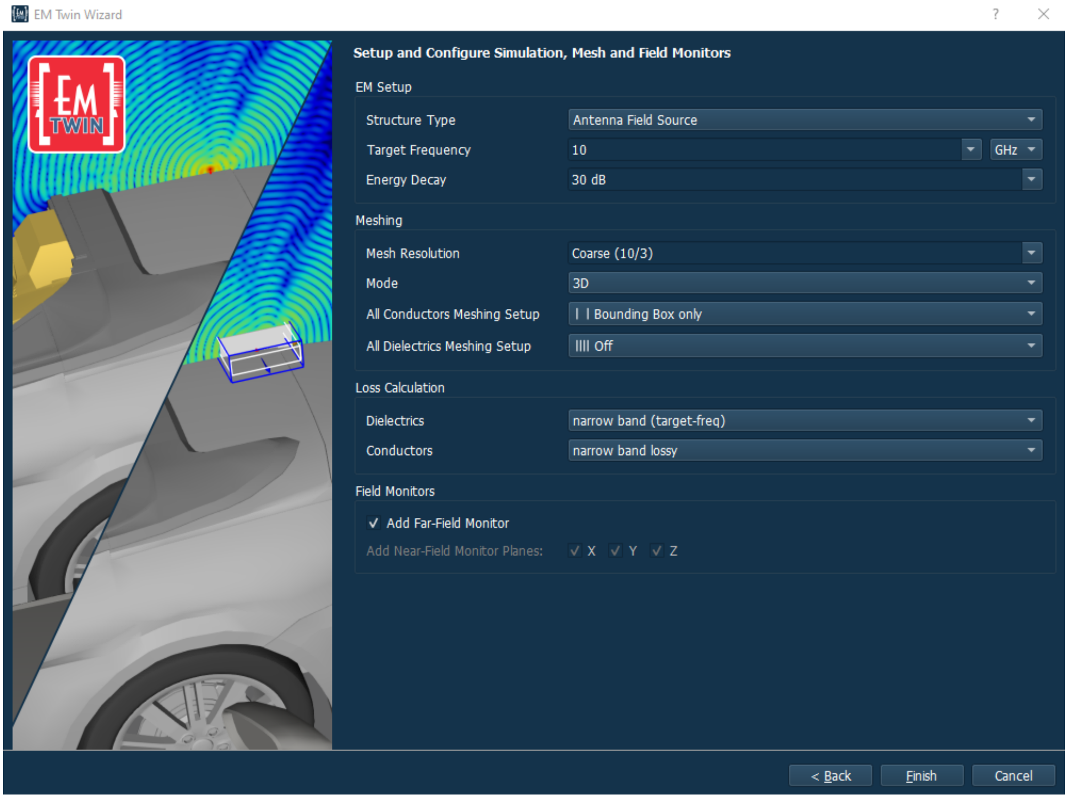


Figure 3.6: Simulation setup wizard

- **EM Setup**
  - **Structure Type**
    Determines distance to boundary, number of time steps, energy decay and resonance estimation order depending on typical simulation problems. It is recommended to keep **Antenna Field Source**.
  - **Target Frequency**
    Used as center frequency in QTEM ports, modulations, narrow band loss models and default frequency in field dumps. It is recommended to use the frequency which is used in the field source file **Surf-Dat**.
  - **Energy Decay**
    Used as simulation end criteria (30 dB recommended for field sources).
- **Meshing**
  - **Mesh Resolution**
    Mesh accuracy (Cells per wavelength / Cells for objects). Can be set to higher resolution for mesh convergence analysis.

  - **Mode**
    Algorithms optimized depending on information about the object types (mainly planar or mainly 3-dimensional). **3D** is recommended for CAD data.
  - **All Conductors Meshing Setup**
    Group mesh level (object relevance for mesh creation) for conductors. **Bounding Box only** is recommended for conductor CAD data.
  - **All Dielectrics Meshing Setup**
    Group mesh level (object relevance for mesh creation) for dielectrics. **Off** is recommended for dielectric CAD data.
- **Loss Calculation**
  - **Dielectrics**
    It is recommended to keep **narrow band (target-freq)**
  - **Conductors**
    It is recommended to keep **narrow band lossy**
- **Field Monitors**
  - **Add Far-Field Monitor**
    Save near field at boundary at target frequency for far field transformation
  - **Add Near-Field Monitor**
    Save near field in planes through center of field source at target frequency

3.1.2 EM-TWIN™ Wizard (Exposure)

This wizard requires basically 3 steps to setup an EM-TWIN™ exposure simulation for a near field source placed beside a phantom model.

1. Setup near field source
2. Setup material and size of the phantom model
3. Setup simulation parameters

Step 1

By clicking the EM-TWIN™ Wizard (Exposure) icon the following window comes up.

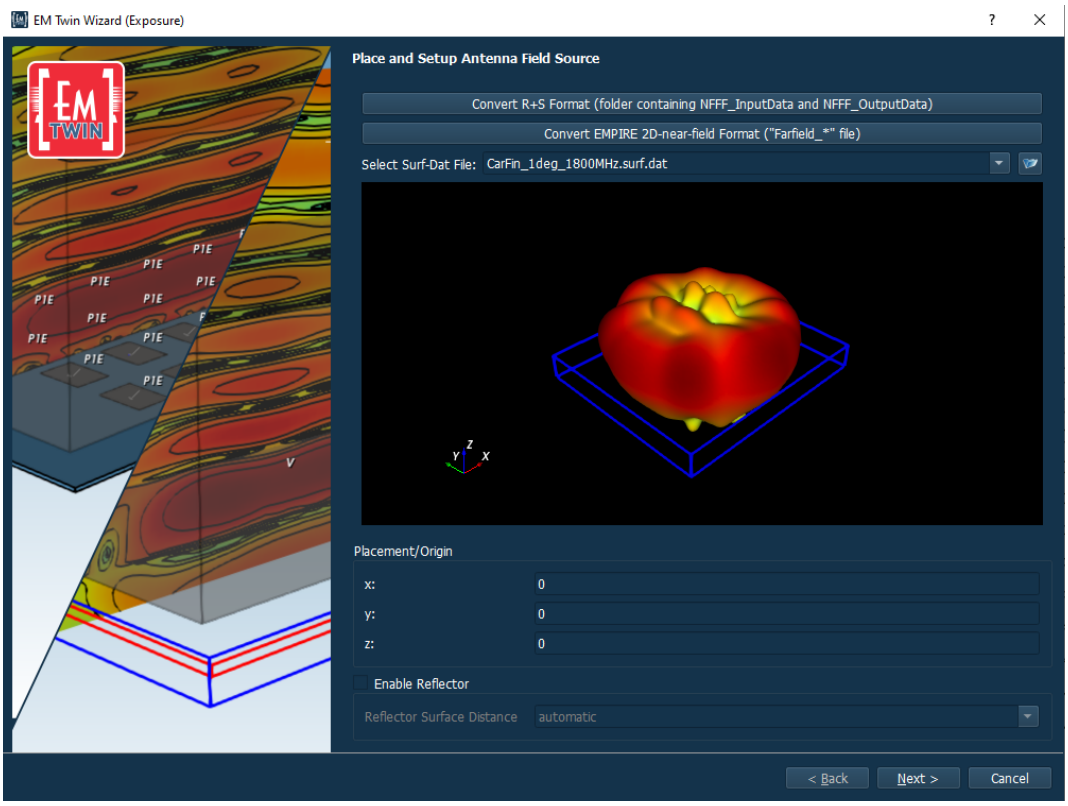


Figure 3.7: Exposure Antenna Field Source wizard

Here, the antenna field source can be selected, configured and placed at the desired location. The field source can be recognized by a file with ending **.surf.dat**. If not already available it can be created from near field data files.

- **Convert R+S Format**
  Select a folder **x** containing near field data from Rohde&Schwarz and create a **Surf-Dat** source file.
  The following data files have to be available in a single folder or following subfolder structure:
  - Mesh files .nas in **x**\NFFF_InputData
  - Configuration file .hyb in **x**\NFFF_OutputData*Frequency*
  - Electric Surface Currents .hmj in **x**\NFFF_OutputData*Frequency*

  - Magnetic Surface Currents .hmm in **x**\NFFF_OutputData*Frequency*
  - Optional 3D radiation pattern .cut in **x**\NFFF_OutputData*Frequency*

- **Convert EMPIRE 2D-nearfield Format**
  Select a boundary near field file (e.g. Farfield_1) from an EM-TWIN™ or EMPIRE simulation and create a **Surf-Dat** source file.
- **Select Surf-Dat File**
  Load a Surf-Dat file with ending **.surf.dat** obtained by conversion or browse for an existing file
- **Placement/Origin**
  Enter a point in the structure (in drawing units) which represents the bottom center of the field source (typically antenna location).
- **Enable Reflector**
  Optionally, specify a reflector and an undersize value in wavelengths. The reflector is an internal part of the field source and is undersized according to the entered distance (0 means reflector size = field source size). The distance can be adjusted from surface to center of the box in order to achieve maximum field at the phantom.

Step 2

As a second step a phantom configuration can be set up.

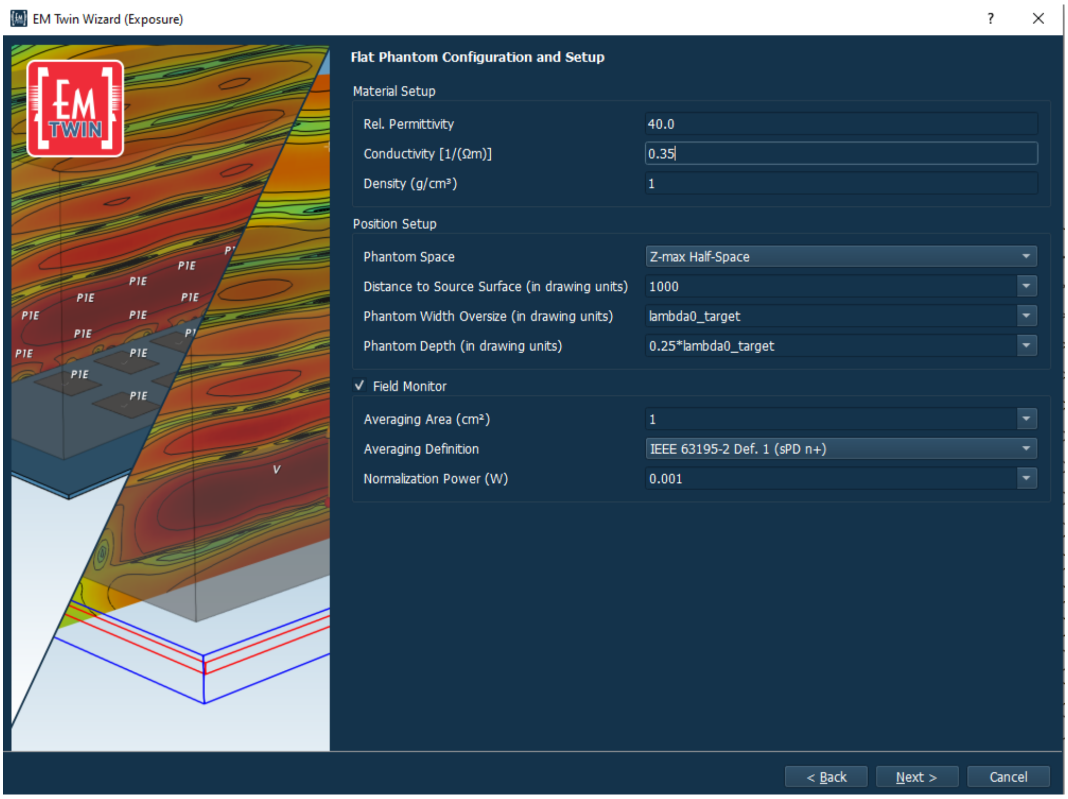


Figure 3.8: Exposure Phantom setup wizard

Material Setup

- **Rel. Permittivity**
  Enter (relative) permittivity of the phantom
- **Conductivity**
  Enter conductivity of the phantom with value in $1/\Omega m$
- **Density**
  Enter mass density of the phantom in g/cm^3

Position Setup

- **Phantom Space**
  Specify which half-space is occupied by the phantom.
- **Distance to Source Surface**
  Specify distance between phantom and source. Enter value in units or in multiples of the target free-space wavelength.

- **Phantom Width**
  Specify size of the additional phantom width. Enter value in units or in multiples of the target free-space wavelength.
- **Phantom Depth**
  Specify thickness of the phantom. Enter value in units or in multiples of the target free-space wavelength.

Field Monitor

- **Field Monitor**
  Disable or enable near field recording for power density evaluation
- **Averaging Area**
  Area for power flow averaging in cm^2
- **Averaging Definition**
  Specify IEEE standard for spatial power density evaluation
- **Normalization Power Weight**
  Scaling value for source power in Watt

Step 3

As the last step simulation parameters have to be set

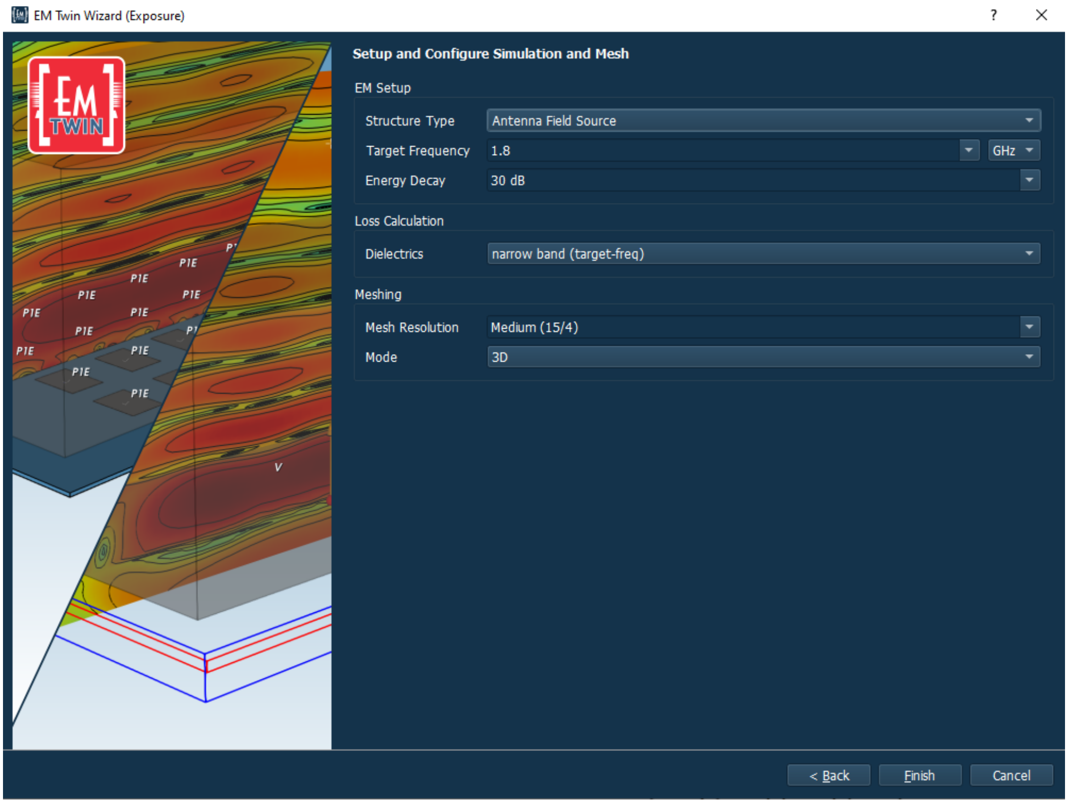


Figure 3.9: Exposure simulation setup wizard

- **EM Setup**
  - **Structure Type**
    Determines distance to boundary, number of time steps, energy decay and resonance estimation order depending on typical simulation problems. It is recommended to use **Antenna Field Source** as structure type for field source applications.
  - **Target Frequency**
    Used as pulse center frequency for sources, narrow band loss models and default frequency in field dumps. It is recommended to use the frequency which is used in the field source file **Surf-Dat**.
  - **Energy Decay**
    Used as simulation end criteria (30 dB recommended for field sources).
- **Loss Calculation**
  - **Dielectrics**
    Set bandwidth for dielectric losses. Determines Debye model order if loss

tangent is specified. It is recommended to apply **narrow band (target-freq)** loss model which has the highest simulation performance.

- **Meshing**
  - **Mesh Resolution**
    Mesh accuracy (Cells per wavelength / Cells for objects). Can be set to higher resolution for mesh convergence analysis.
  - **Mode**
    Algorithms optimized depending on information about the object types (mainly planar or mainly 3-dimensional).

3.2 Elements of the User Interface

The simulation workflow implemented in the user interface can be described as:

- Create a model with the aid of the EM-TWIN™ Wizards
- Optionally add additional objects, sweep variables
- Check model, mesh, simulation parameters, field monitors, ...
- Run the simulation (single simulation, batch queue, parameter sweep or optimization)
- Evaluate results (near field, far field, SAR, sPD, ...)

The different elements of the user interface are depicted in the following:

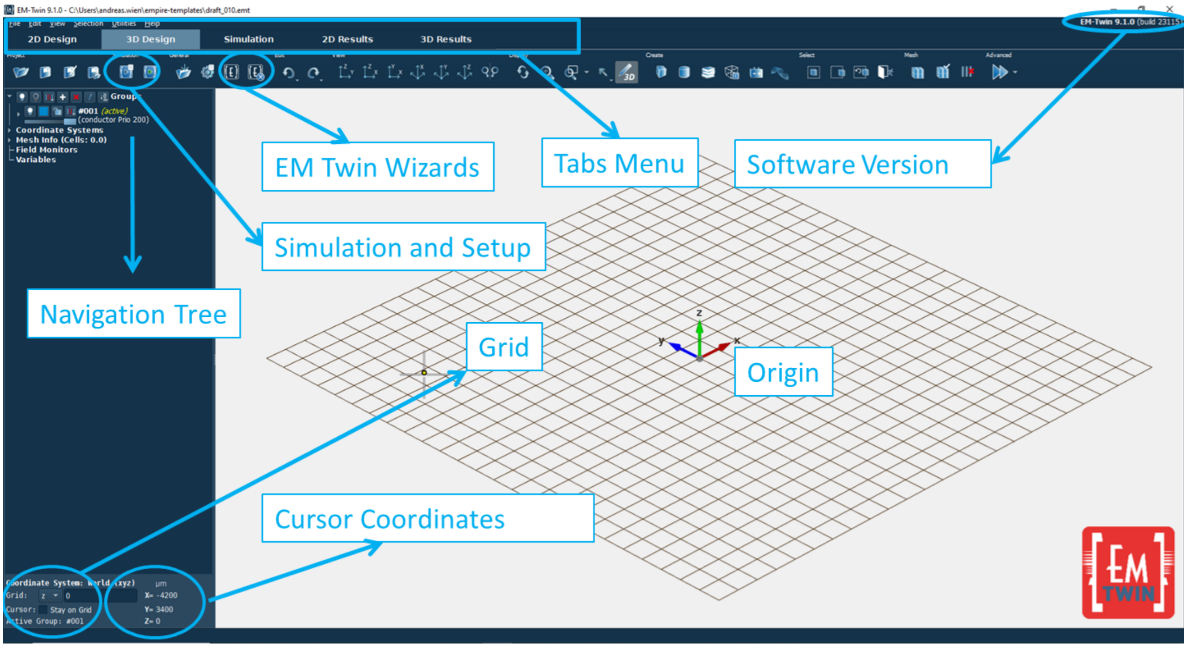
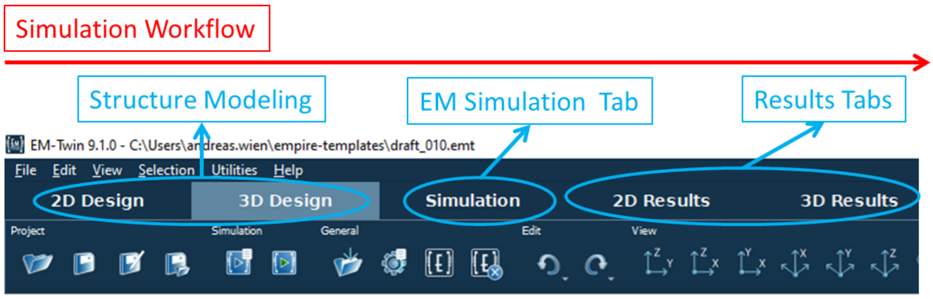


Figure 3.10: Different elements of the user interface

Navigation Tree

- **Groups**
  In the group tree you have access to the structure and source elements used in the simulation model. These elements can be edited or new groups and elements can be added.
- **Coordinate Systems**
  If different coordinate systems are defined they can be controlled here.
- **Mesh Info**
  Information on current mesh number and cell spacing is available here.
- **Field Monitors**
  Field monitors are listed and can be edited or added in this part of the user interface
- **Variables**
  Object's coordinates as well as values of properties, e.g. permittivity, may be parametrically defined. The navigation tree contains also the variable list with the sweep parameter options.

Tabs Menu



The tabs in the Tabs Menu are organized following the simulation workflow. Using these tabs you can switch between different functionalities of EM-TWIN™.

- **2D Design**
  The 2D Design tab activates planar drawing planes where the model can be edited in 2D which is useful for exact positioning and checking. In addition the current mesh can be overlayed.
- **3D Design**
  The default 3D Design tab provides a grid to enter the model in 3D view. The cursor snaps to grid coordinates and important object points for a quick and intuitive creation and modification of objects. Grid properties can be accessed in the Context Menu Window (right click) and local coordinate systems (LCS) can be derived from object faces.
- **Simulation**
  After starting the simulation, the Simulation tab is active and the system energy will be displayed during the simulation. The Simulation tab is used for advanced simulation control, e.g. to queue or distribute jobs, or run variable sweeps and optimizations. Simulation details and statistics are printed in a log window. By default the energy decay versus time is displayed.

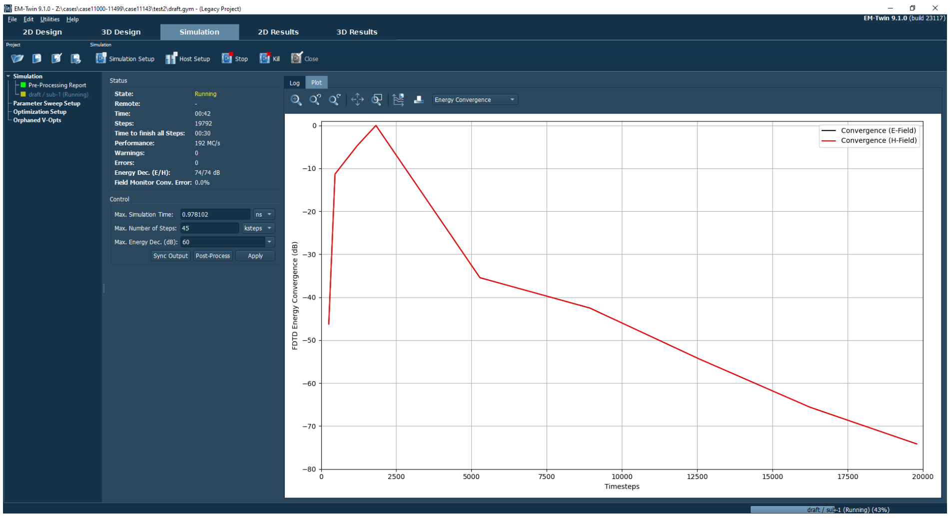


Figure 3.11: Simulation window - energy evolution

- **2D Results**
  Plots, like voltage and current time signals, frequency dependent parameters as well as far field cuts can be displayed in this tab. The 2D Results Tab shows result plots in different predifened windows (which can be customized).

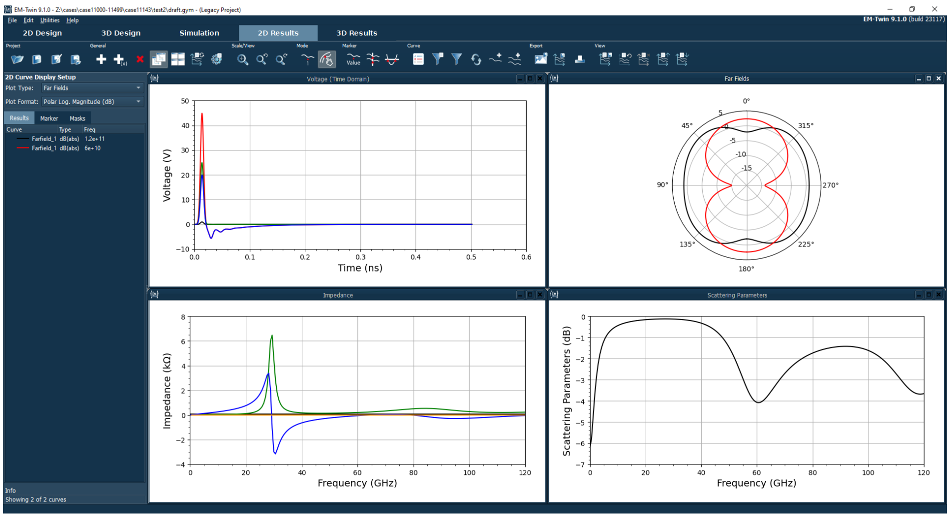


Figure 3.12: 2D Results window

- **3D Results**
  A rendered view of the geometry, mesh and boundaries is obtained in the 3D Results tab which is also used to display field animations after simulation. For the purpose of a near field and far field plots, field monitors have to be defined.

After simulation the field distribution and far fields can be visualized together with the structure if the respective field monitors have been set up. For more details see Chapter 4.4.

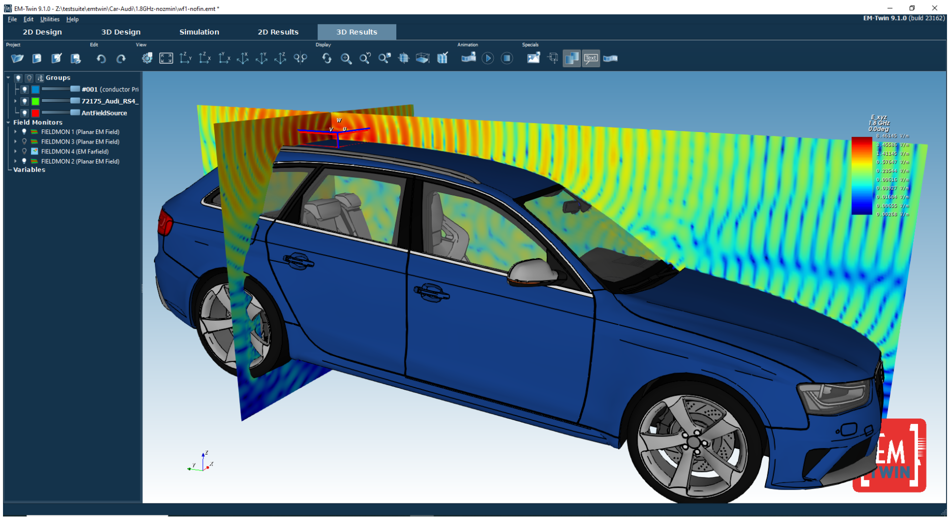


Figure 3.13: 3D Results window

Hint

More information about the user interface is available in **EMPIRE-Manual.pdf - Part II: Empire User Interface**

3.3 Advanced EM-TWIN mode

By default only a subset of user elements and functions are available. In order to obtain the complete functionality (e.g. 2D Library elements, ports, etc.) it can be activated in **Help - GUI Configuration - EM-Twin Settings**

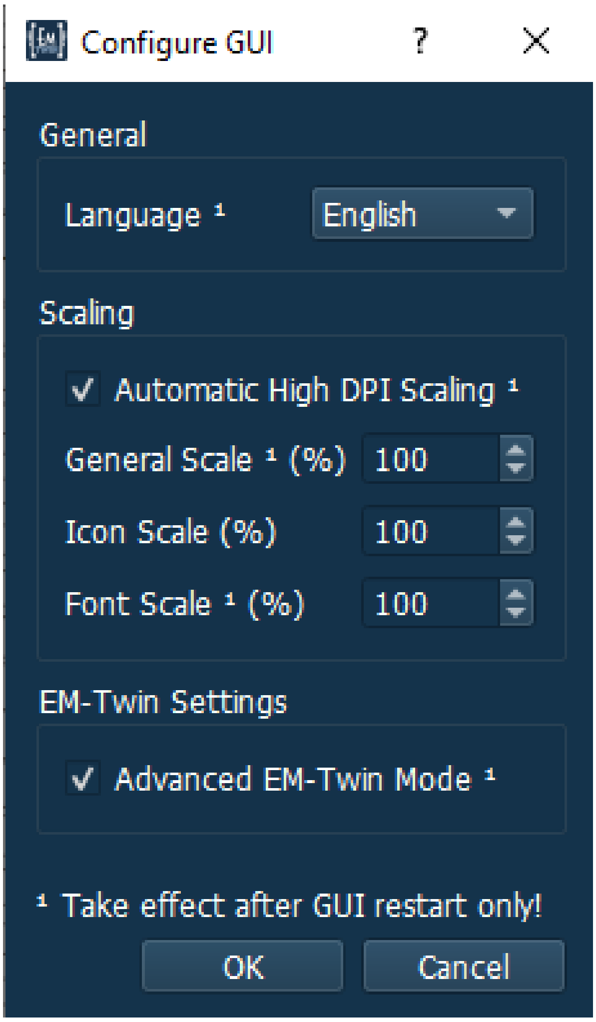


Figure 3.14: Advanced EM-Twin Mode

4 Structure Modeling

This chapter provides an overview of EM-TWIN™ basic components and workflow. The various aspects of the different components of the software are discussed as they are encountered in the simulation workflow.

4.1 CAD Import

EM-TWIN™ supports several import methods.

Without simplification the CAD data is imported using

- the EM-TWIN™ Wizard **CAD Import**
- the import button
- the menu **File - Import - 3D Solids**

In case of large CAD data an import tool is available which can reduce the amount of data to be imported using

- the EM-TWIN™ Wizard **Large CAD Import Tool**
- the menu **File - Import - Advanced - 3D Solids - Run Tool for Large Data Import**

4.1.1 3D Import Formats

The following described 3D CAD formats are supported and if necessary some import options can be set in advance in the **Editor Options - Import** menu.

The following formats are supported without additional license:

- **Collada** (ending .dae)
  Interchange format for interactive 3D applications, based on 3D polygonal meshes.
- **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 EM-TWIN™

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

Hint

More information about import and export formats is available in **EMPIRE-Manual.pdf - Chapter 17: Import and Export**

4.1.2 Import Options (Facets)

While importing Polygonal Mesh (Facets) CAD-files, e.g. *.stl, the following dialog appears.

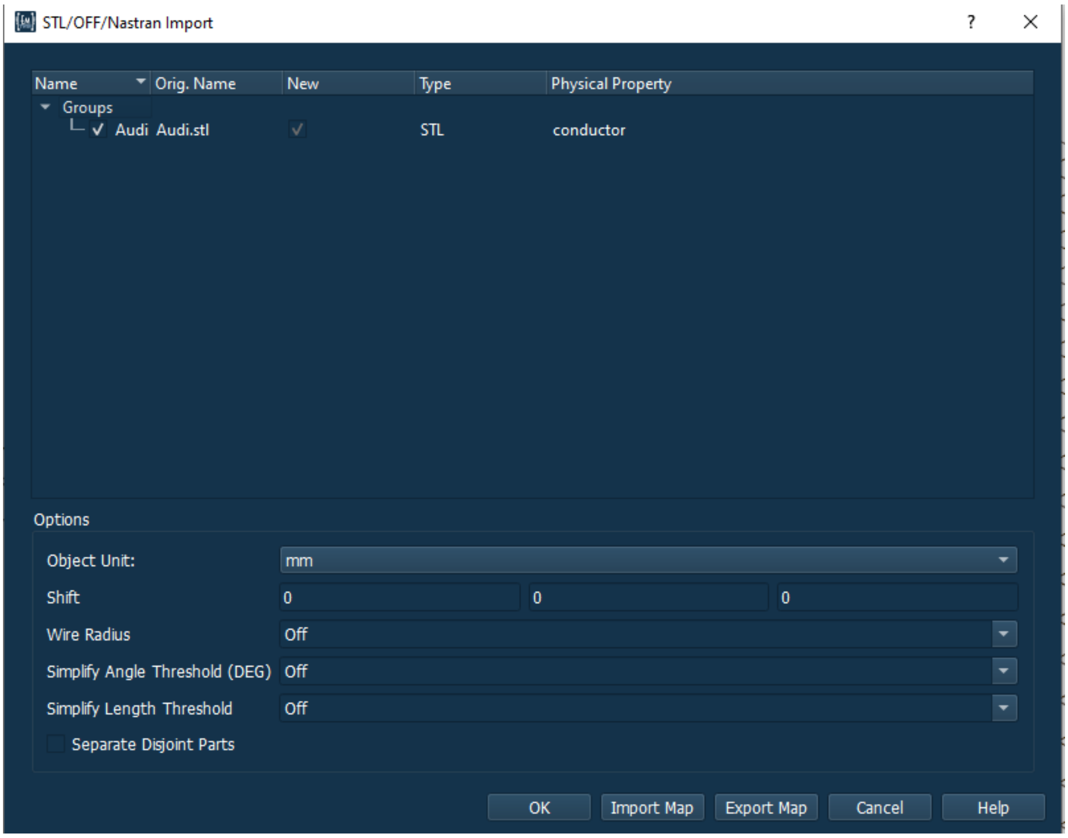


Figure 4.1: Facets import options

At the top the group organization and their physical property can be set.

- **Unit** - To be adjusted if drawing units are different
- **Shift** - To be set if origins are different
- **Simplify** - Reduce complexity of solids to speed up import and drawing time. Either an minimum angle or length can be specified for the resolution
- **Separate Disjoint Parts** - 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**

4.1.3 Import Options (BREP)

While importing Boundary Representation (BREP) CAD-files, e.g. *.sat, the following dialog appears.

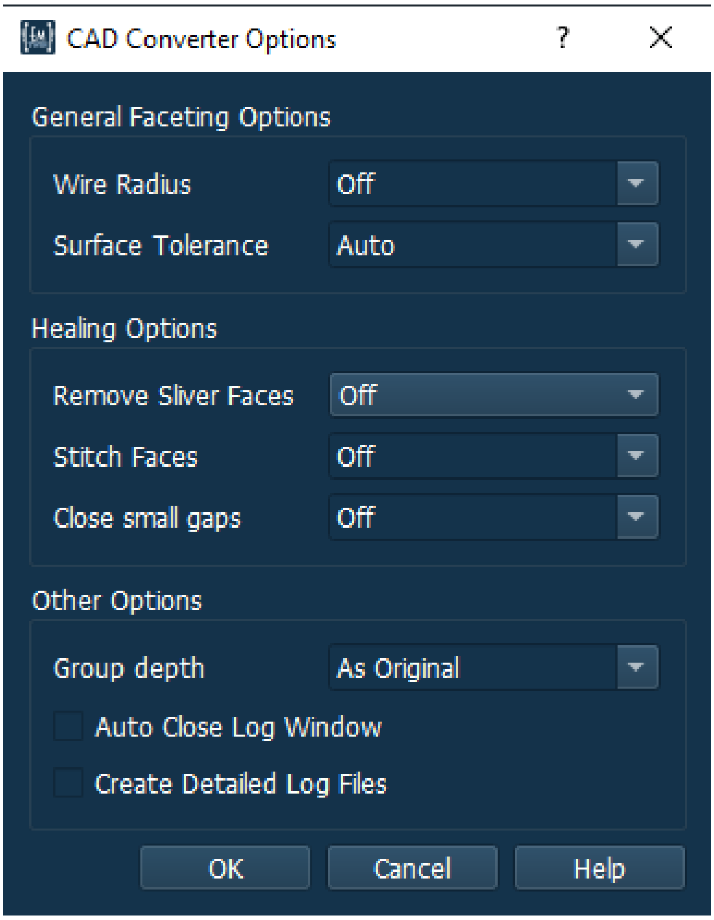


Figure 4.2: BREP import options

Its settings should be checked and adjusted:

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

should be smaller than the minimum feature size and bigger than the maximum gap expected to be stitched in the model.

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

4.1.4 Large CAD Import Tool (LCI)

An import tool is available which is dedicated to vehicle models. The built-in algorithms reduce the complexity of the CAD data and generate equivalent models without impairing the electrical behavior for the targeted EM simulation. Depending on the selected simplification method the resulting dataset is strongly reduced in size to be handled easily by 3D EM simulation.

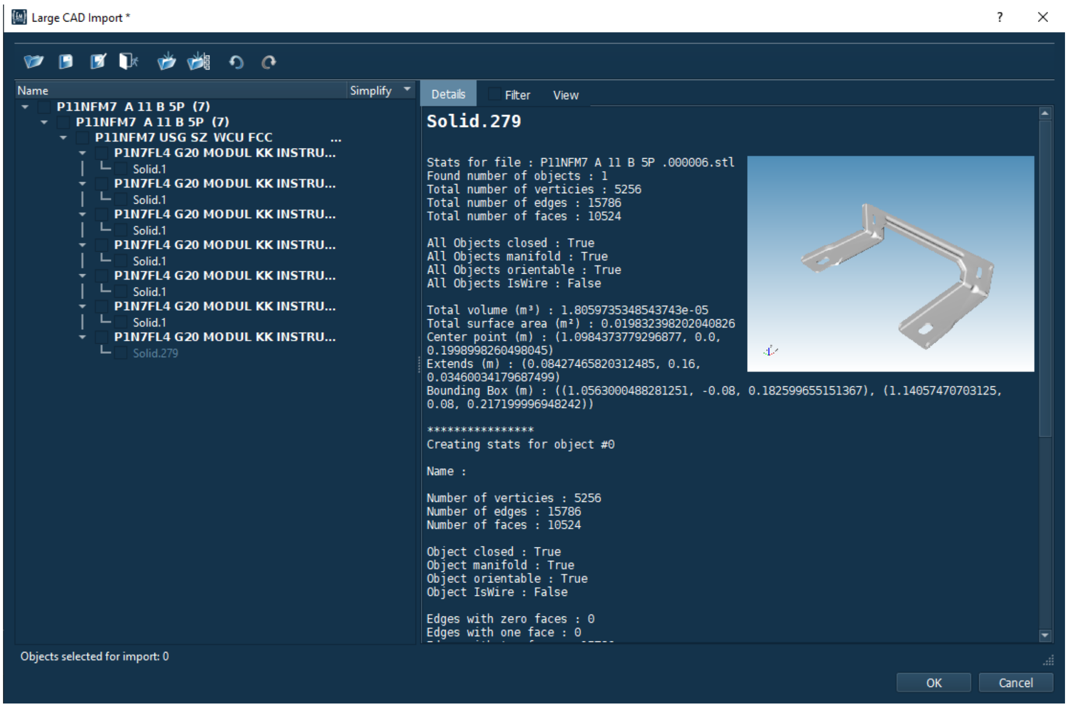


Figure 4.3: Large CAD import tool

Elements of the LCI

- **Top Icons** Here an import configuration can be saved, loaded or reset. With the icon  the CAD file can be imported into the LCI tool for configuration.
- **Name** After import the elements of the CAD data are listed in hierarchical order.

  - Objects can be selected for import with a checkmark
  - Objects can be marked with a left mouse click to be displayed in the window on the right.

- **Simplify** With a right click on the marked objects a simplification method can be selected. This method is then displayed in this column
- **Details** For the marked object statistics and a preview will be displayed
- **Filter** A filter can be applied for the names in the tree on the left to reduce the number of elements. So elements with common name (e.g. Screw) can be gathered and simplification methods assigned at once.
- **View** In this tab the simplification of the marked objects will be displayed

LCI Example

As an example for simplification a screw element is marked in the picture below with different simplification levels assigned to.

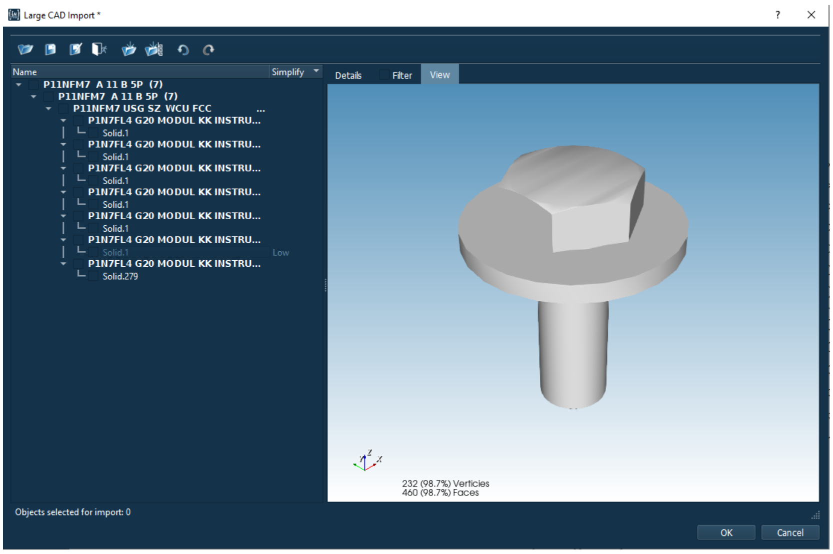


Figure 4.4: Simplification level: Low

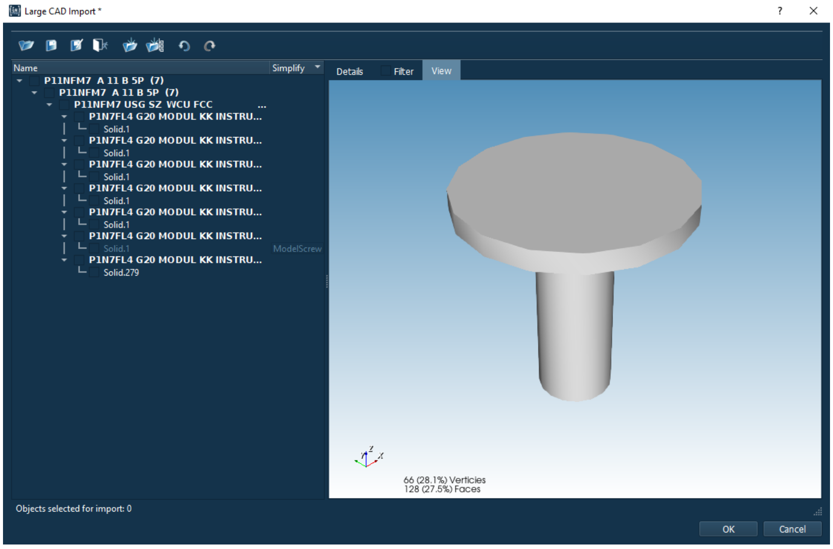


Figure 4.5: Simplification level: ModelScrew

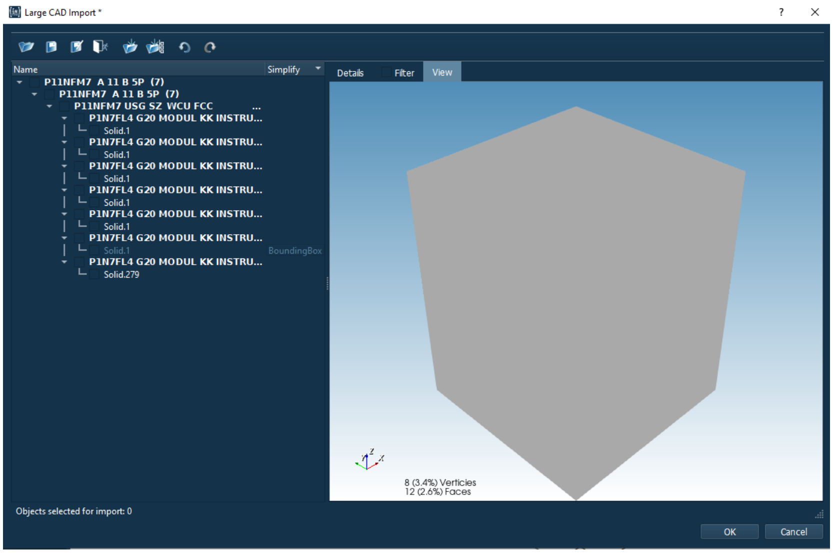


Figure 4.6: Simplification level: BoundingBox

4.2 Create a model

Objects (Boxes, Polygons, Wires, Solids) with common properties are organized in **Groups**. It is recommended to create a group, set its electric property, create the objects which belong to this group and continue with a next group. Certain objects like polygons, wires and ports can take advantage of the group height.

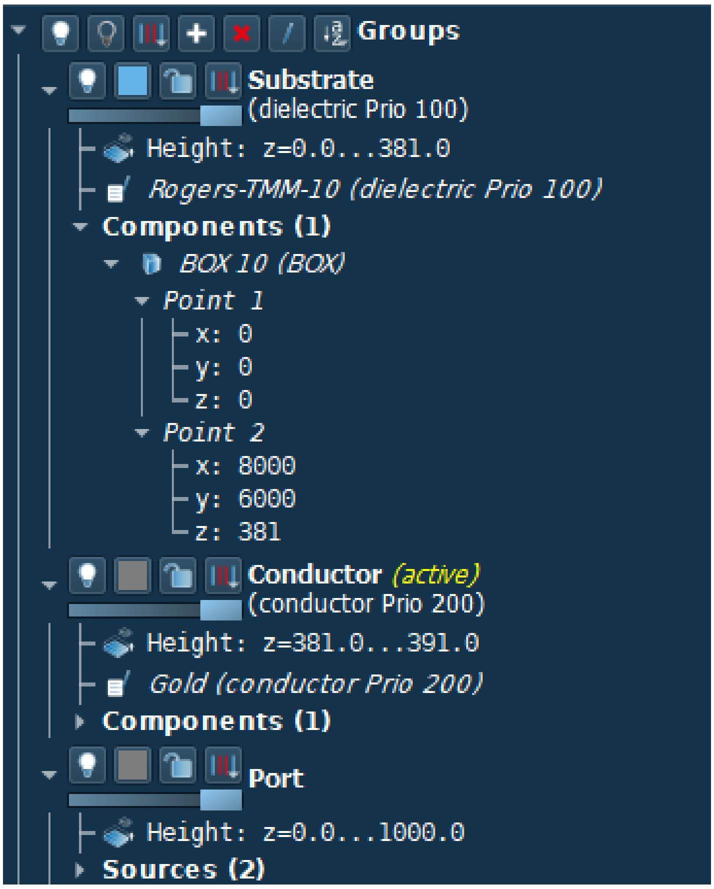


Figure 4.7: Group list

EM-TWIN™ features a wizard for data import. Imported 3D data (STL, ACIS, STEP, ...) create groups automatically and only their properties have to be defined.

4.2.1 Groups and properties

Objects with different properties are organized in separate groups. Their properties can be either physical (e.g. material), geometrical (e.g. height) or functional (e.g. mesh hint). The groups are controlled with the list in the navigation tree. Here, name, color and style can be set to distinguish the groups and each of them can be locked or hidden to prevent modification of objects.

New objects are always created in the active group which is displayed by in the bottom line of the GUI. A new group is automatically marked as active. To activate another group press right mouse button on the group name and select ”Set Active”.

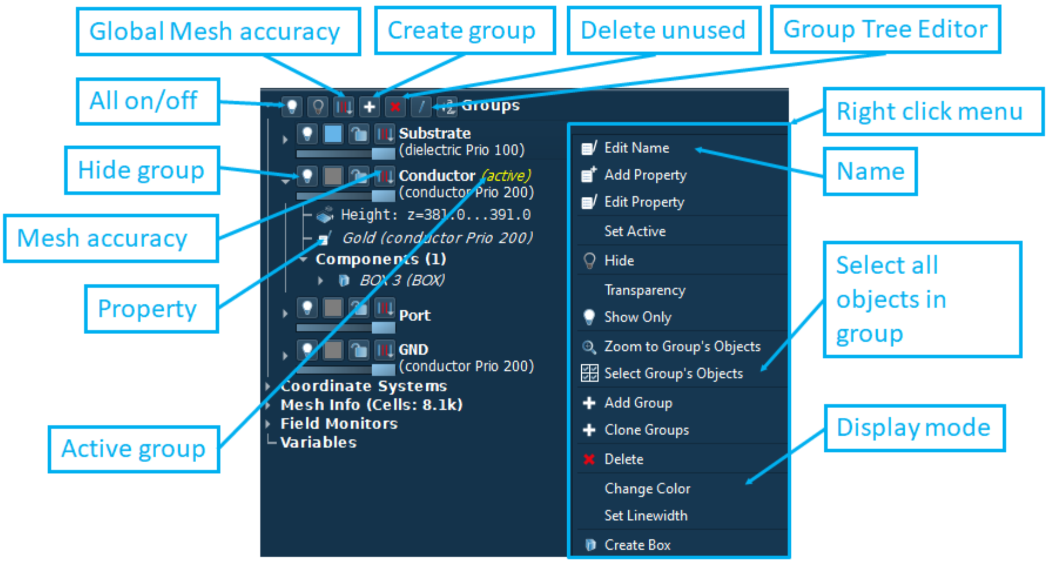


Figure 4.8: Group Organization

The group height can be used to assign box or polygon objects a certain perpendicular extension or the transverse position of library objects.

All objects in a group inherit its properties by default. In some cases a group can have more than one property (e.g. conductor and mesh hint).

Furthermore, a meshing mode can be selected for each group in order to classify the significance of the group's objects to the simulation as in following:

- Default mesh level, mesh objects edges and interior
- Mesh only objects edges
- No meshing for objects in this group
- Mesh object edges and fine interior
- Ignore edges, mesh only interior
- Mesh only gaps

Hint

More information about groups is available in **EMPIRE-Manual.pdf - Chapter 14: Groups**

When pressing the add or edit property button in the group list the **Object Property Editor** (Fig. 4.9) is activated. Several predefined properties can be selected and assigned to the group objects. Material properties should be selected using **Assign Material properties** where a comprehensive database of materials (Dielectric, Conductor and Absorber) is available and a user database can be created.

A certain priority number can be assigned to each material property, which defines the prevalent property in case of intersecting objects. A higher priority number means also higher priority for the edited material.

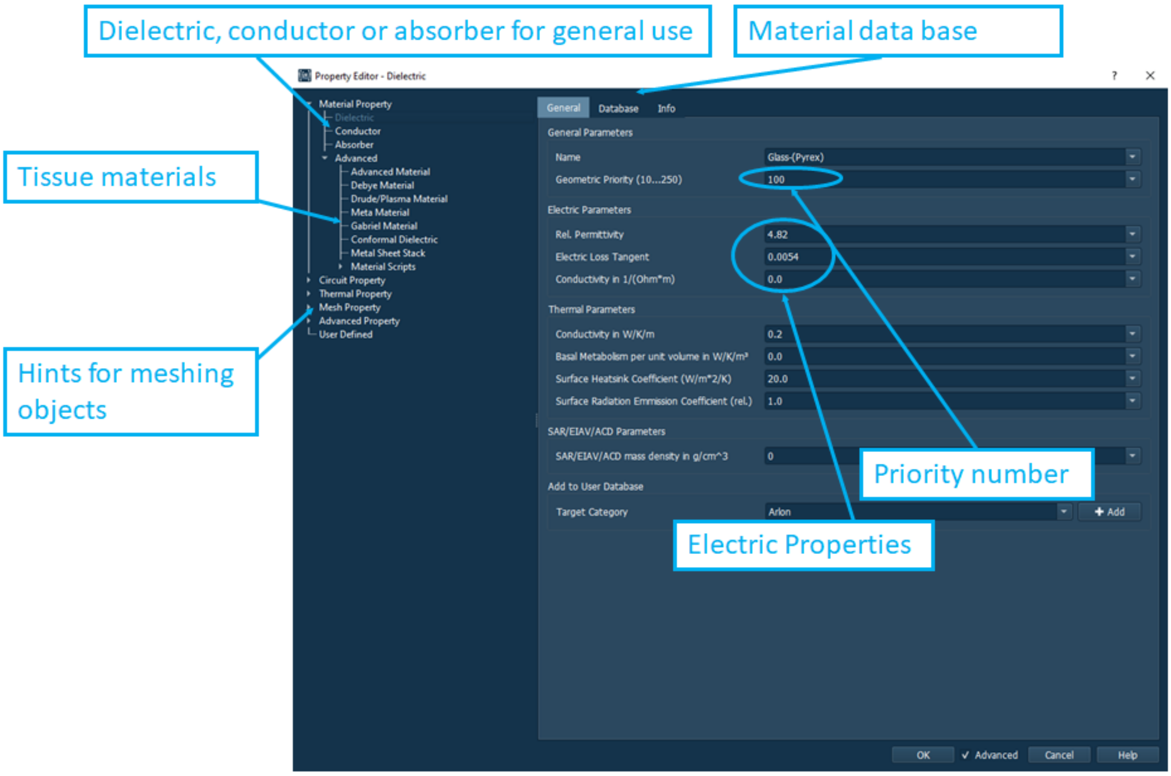


Figure 4.9: Object Property Editor window

Advanced properties contain special models or excitation features.

Hint

More information about properties is available in **EMPIRE-Manual.pdf - Chapter 18: Object Properties**

4.2.2 Create objects

Objects are generated with the "Create" menu buttons and are available in the group list. Object modification is possible after selection.

In EM-TWIN™ basic objects are Boxes, Polygons and Solids. Their edges or corners are represented by handles (visible after selection) which can be selected for modifications (stretch, move...). After creation, these objects are available in their **Groups** list in the navigation tree on the left.

- **Box** objects can be defined by pressing the **Create Box** button and following the instructions. Another option is to create an arrow, which spans the cross section and pressing the button afterwards. In this case, the group height has to be defined perpendicular to the current view, as it sets the depth of the box by default.
- **Poly**(gon) objects can be defined by pressing the **Create Poly** button and following the instructions. They can also be created by entering a sequence of points (Shift + left click) and pressing the button afterwards. A single point Poly with a radius represents a circular cylinder, while for multiple points the radius acts as the rounding of the corners. Imported layouts are always processed as Polys.
- **Solid** objects are usually the result of Boolean operations or non-orthogonal rotations applied to objects. This object type is also obtained after exploding a library object and if 3D data in STL format is imported from other sources.

More complex objects can be found in the object library which is available by pressing **Create Library Object** (Fig. 4.10), entering coordinates and adjusting **Parameters**. If these objects have to be modified by, e.g. Boolean operations, they have to be converted first to solids or polygons by the **Explode** operation.

By default, only a subset of objects are available in EM-TWIN™. The complete range of objects are available using the **Advanced** mode, see Chapter 3.3.

Figure 4.10: View of some Library Object

After creation, all objects are available in the **Group** list and their properties can be edited. They may be modified, e.g. to adjust the co-ordinates or to add variables (e.g. r=100+x).

Hint

More information about object creation is available in **EMPIRE-Manual.pdf - Chapter 15: Object Creation**

4.2.3 Selections and Operations

The operation toolbar is dynamic and displays available possible actions depending on current selections.

Hint

Object Selection:

- move the cursor to an an object's edge or face. Click the left mouse button.
- Select multiple objects: Use functions Select overlapping outside/inside buttons.
- Select all (visible objects): Ctrl a
- Open group list, click on object to be selected
- Select all group objects: right click on group, button "Select Group's Objects"

More operations can be accessed through the **Advanced** button. All buttons can also be accessed in a menu with right mouse click in the design window.

After clicking on one of these buttons, a dialog box appears with a request to enter points, arrows, numbers, etc. to complete the desired operation.

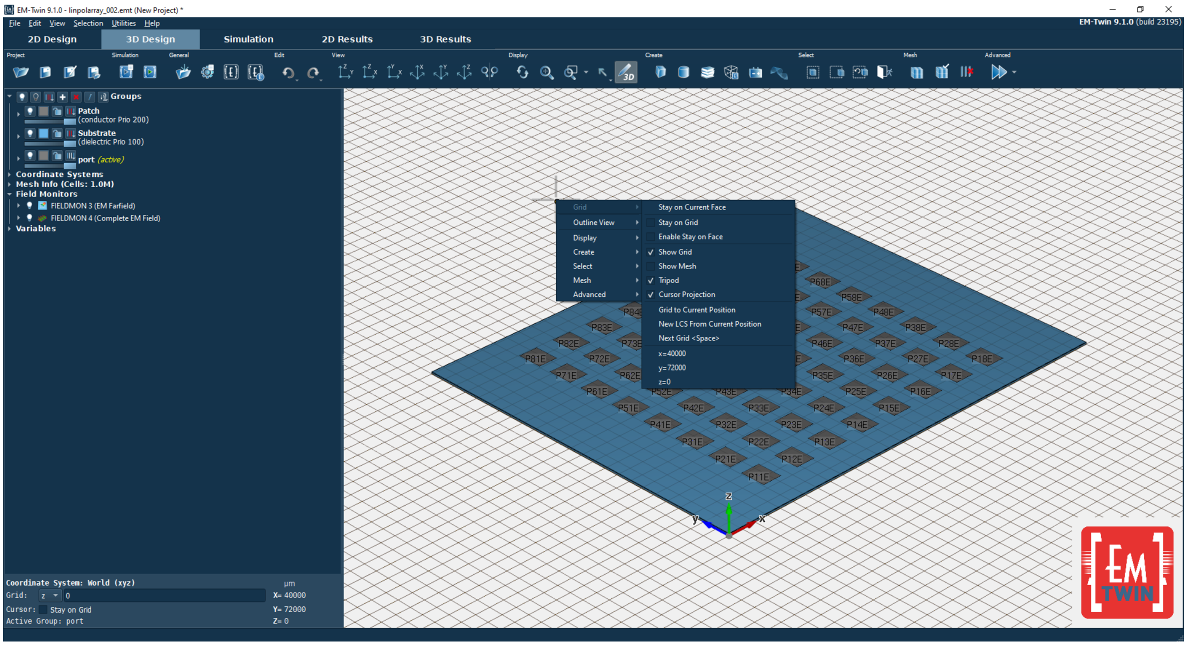


Figure 4.11: Toolbar and right click menu

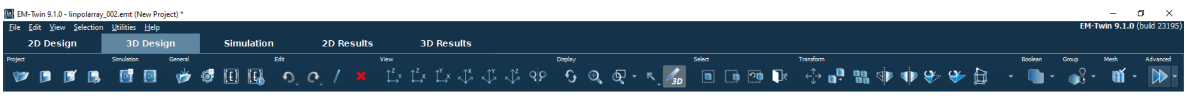


Figure 4.12: Toolbar after box selection

Some operations, like move, copy or stretch, can be performed by mouse input. After clicking the object is left highlighted and handles are visible which represent the vertices of polygons or edges of boxes.

Non selected handles (red) can be activated (to green) by clicking the left mouse button on them. By moving the cursor to one of the green handles the following actions can be performed in the drawing area:

- stretch active points (press and drag left mouse button)
- copy object (press and drag middle mouse button in 2D Design)
- move object (press and drag right mouse button in 2D Design)

Further, points can be inserted, replaced or deleted.

Hint

Short cuts

- z/arrow: zoom extends/region
- u: undo
- r: redo
- p arrow: select overlapping
- e arrow: select enclosed
- Ctrl a: Select all
- Ctrl x: Cut to clipboard
- Ctrl c: Copy to clipboard
- Ctrl v: Paste from clipboard
- Space bar: Swap grid (3D Design)

Hint

More information is available in **EMPIRE-Manual.pdf - Chapter 13: Operators, Selections and Shortcuts**

4.3 Define a field source

After object definition excitations have to be defined. In EM-TWIN™ the structure is excited by an antenna field source which can be created using the wizard or using the **Create Antenna Field Source button** . The structure is excited by a modulated pulse exciting electric and magnetic currents on the surface of the field source box.

4.3.1 Field Source Location and Orientation

Hint

A field source is a fixed-size box (as defined by the imported Surf-Dat file) and 3 points in space which determine the box orientation. Because the source has to be aligned to the World Coordinate System (WCS) before simulation the complete structure has to be rotated if it is constructed on a curved surface.

The source elements requires the input of 3 points in space which define a tripod $\vec{u}$, $\vec{v}$, $\vec{w}$.

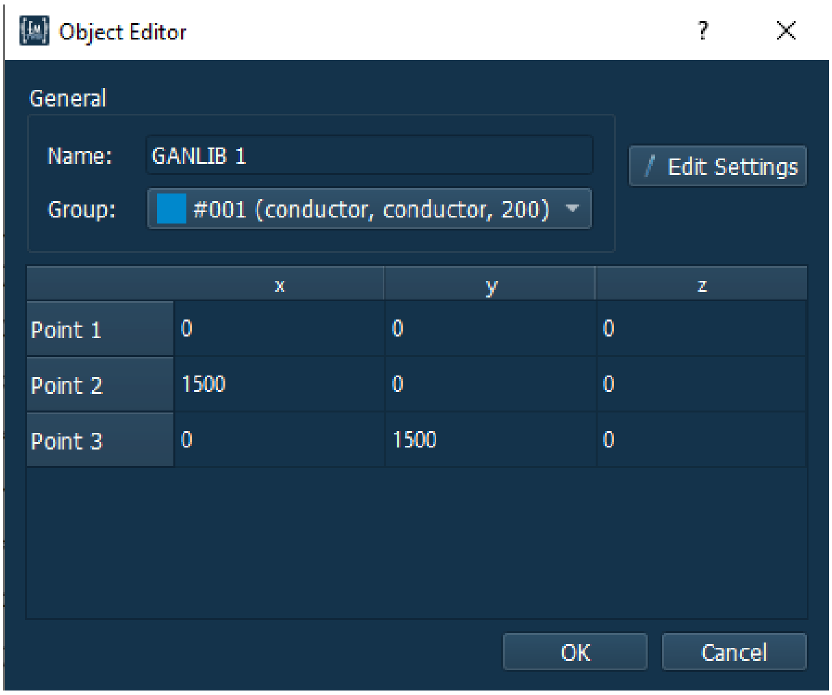


Figure 4.13: Object Editor for Antenna Field Source

- **Point 1** Bottom center of field source and origin of the tripod $\vec{u}$, $\vec{v}$, $\vec{w}$
- **Point 2** Tip of $\vec{u}$ vector

- **Point 3** Tip of $\vec{v}$ vector

The $\vec{w}$ vector is obtained from the cross product $\vec{u} \times \vec{v}$.

4.3.2 Field Source Display

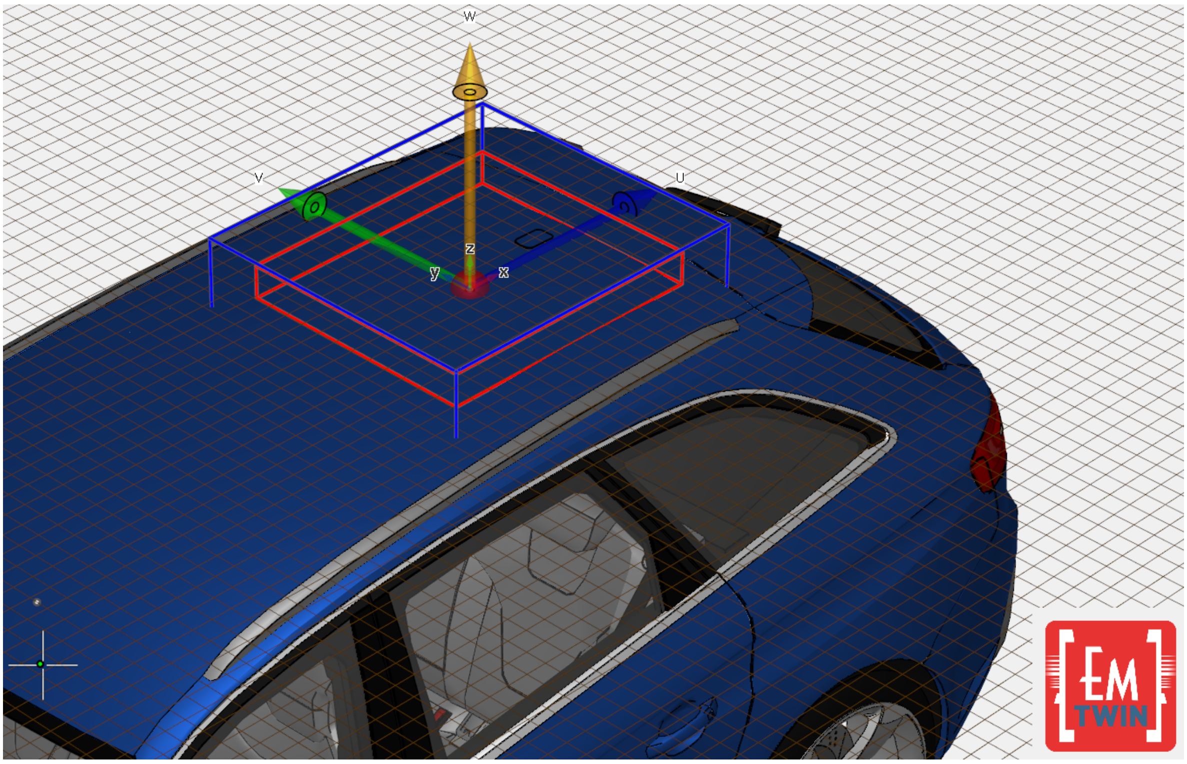


Figure 4.14: Antenna Field Source and uvw tripod after rotation

Hint

The antenna field source size is displayed by the red box. The power measurement surface is indicated by the blue box. Its oversize value can be adjusted in the library editor. If a reflector is enabled it is displayed by a second red box inside the red source box.

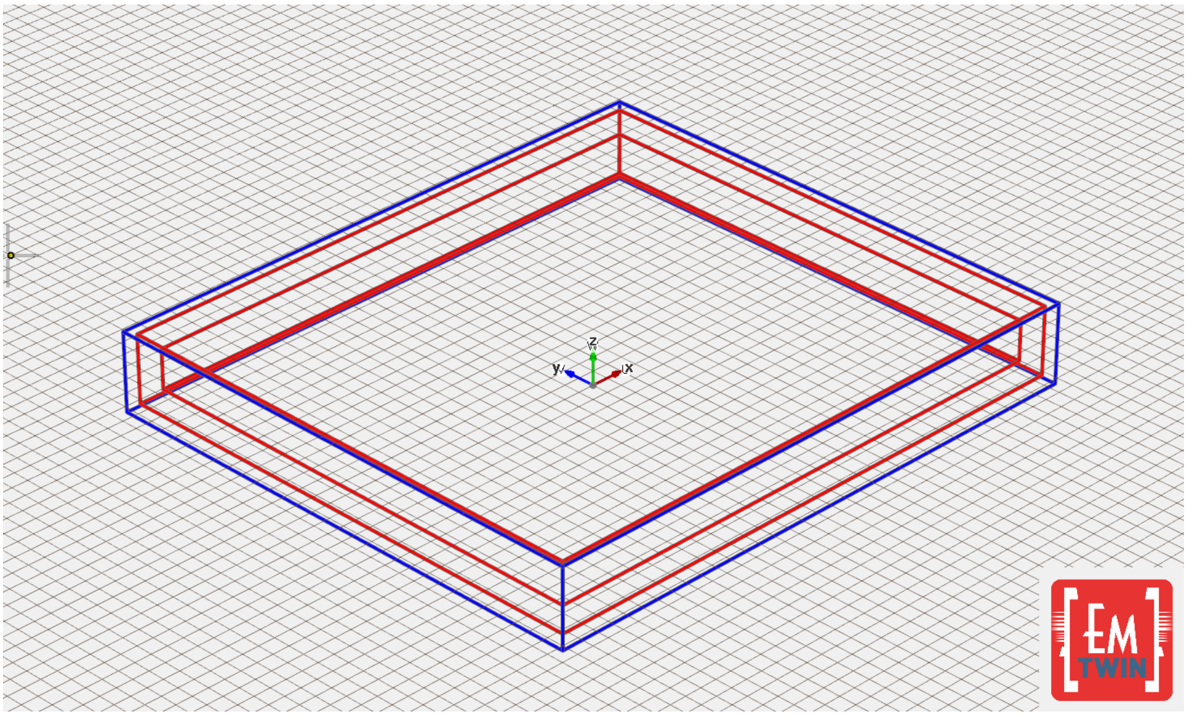


Figure 4.15: Antenna Field Source with enabled reflector inside

4.3.3 Field Source Parameters

Figure 4.16: Library Editor for Antenna Field Source

- **E+H-Field Surface File**
  Load a Surf-Dat file obtained by conversion or browse for an existing file.
- **R+S Format**
  Select a folder **x** containing near field data from Rohde&Schwarz and create a **Surf-Dat** source file.
  The following data files have to be available in a single folder or following subfolder structure:
  - Mesh files .nas in **x**\NFFF_InputData
  - Configuration file .hyb in **x**\NFFF_OutputData*Frequency*
  - Electric Surface Currents .hmj in **x**\NFFF_OutputData*Frequency*
  - Magnetic Surface Currents .hmm in **x**\NFFF_OutputData*Frequency*
  - Optional 3D radiation pattern .cut in **x**\NFFF_OutputData*Frequency*
- **EMPIRE 2D-near-field Format**
  Select a boundary near field file (Farfield_) from an EMPIRE simulation and create a **Surf-Dat** source file.
- **Port Number**
  Specify port number, e.g for multiple excitations
- **Port Excitation**
  Switch On or Off for, e.g. neglection
- **Power Measurement Oversize**
  Distance for power measurement in free-space wavelength
- **Reflector Box Size**
  Relative undersize for a reflector box in wavelengths. If enabled, the reflector is visible as a second red box inside the field source box. If 0, the reflector size is equal to field source size. The distance can be adjusted from surface to center of the box in order to achieve maximum field at the phantom.
- **Disable Sides**
  If the source is placed on a metal surface, the respective side of the source should be disabled (typically zmin).
- **Delay**
  Can be used in multiple excitations as a phase shift at the target frequency
- **Options**
  Scale factors for electric and magnetic surface currents
- **Far Field Center Offset**
  In case of a far field source the center can be placed relative to the source origin

- **Far Field Rotation Euler angles**
  In case of a far field source the orientation can be set relative to the near field source orientation
- **Rotate for Simulation**
  If the field source is not aligned to the world coordinate system, it has to be rotated including the complete structure. Using this button automatically rotates the complete model so the field source is properly aligned.

Hint

More information about field sources and other ports is available in **EMPIRE-Manual.pdf - Chapter 20: Sources**

4.4 Define field monitors

The EM-TWIN™ Wizards create some default field monitors. If near additioanl field plots or radiation patterns should be generated, they can be defined in **Field Monitors** by pressing the **Create Field Monitor** button in the toolbar (Fig. 4.17).

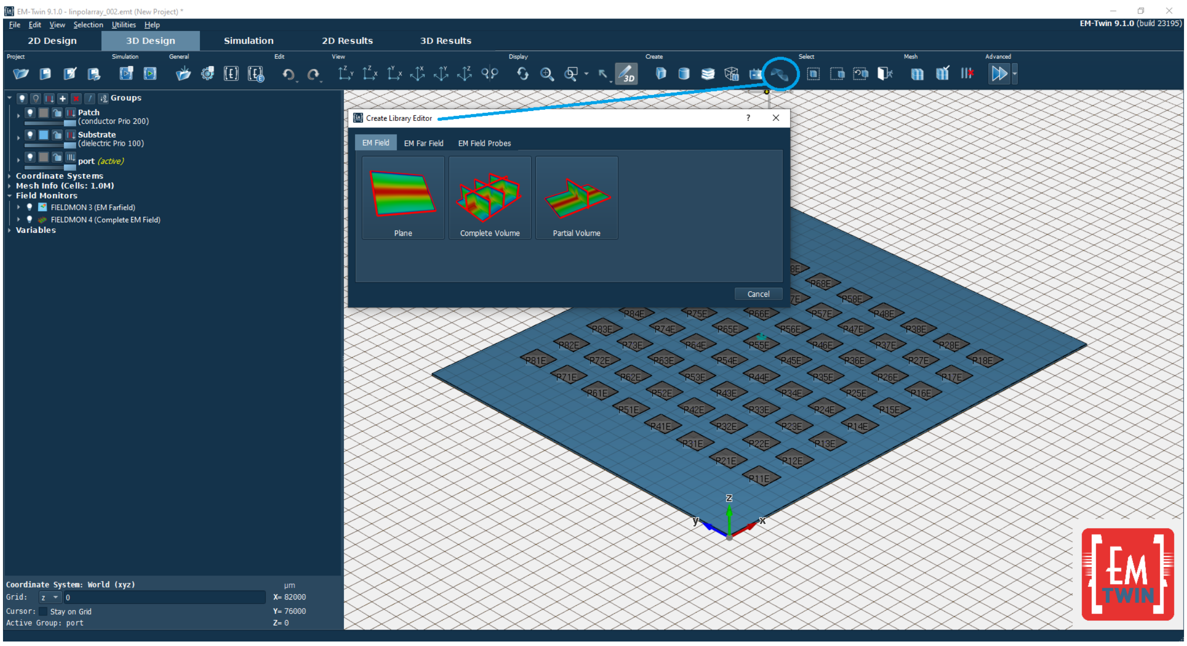


Figure 4.17: Field Monitors

In **EM Display** section, near fields are recorded in a defined area and can be visualized in the Display 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. This monitor displays only E or H field.
- `Complete EM Field`: specifies the complete simulation domain for field storage (memory intensive). For the 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 the display, several planes may be defined and varied with a slider.

In the **EM Far field** section, near fields are recorded on a box (Huygens Surface) defined by the boundaries or by specified by the user. The far field is obtained in the post processing by a transformation, which requires homogeneous space all around this surface. As soon as this monitor is used, additional space is added around the structure automatically by the meshing algorithm. If the automatic meshing is disabled and some material (dielectric or metal) is extended to the boundary this can lead to unphysical results and warning messages.

The far field pattern can be visualized in the **3D Result** tab (together with the structure) or plotted in the **2D Result** tab if far field angle cuts are defined.

In **EM Probes** section, near fields are recorded in defined points or along defined paths and can be visualized with the plotting engine.

- `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.

4.4.1 Near and Far Field Setup

The parameters for a near and far field record are very similar (Figures 4.18–4.19). Moreover the number of frequency points or time steps used for field recording has to be carefully selected, as it will affect the amount of memory needed while the simulation is performed. It is possible to enter single frequency points or a sequence of equidistant frequency points.

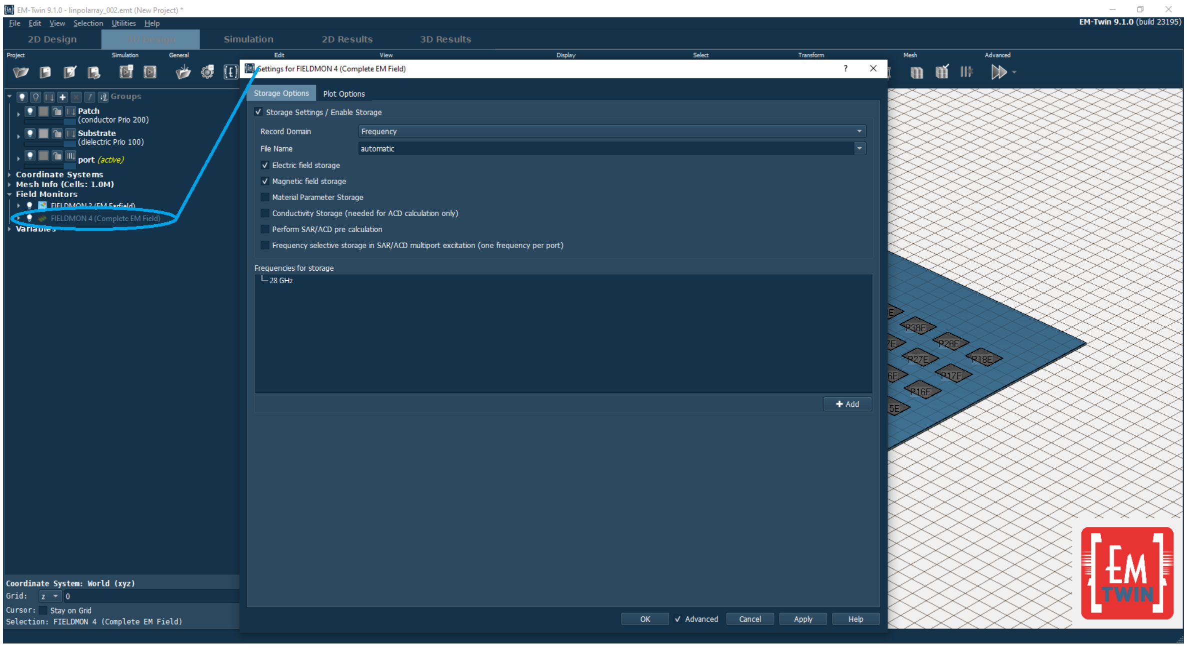


Figure 4.18: Near Field Setup Window

By default the far field calculation will be performed as a directivity 2D cut in the $xz-$plane ($\phi = 0°$), $yz - plane$ ($\phi = 90°$) and as a 3D plot with an angle step of 5°. Other results or options can be obtained by adjusting the parameters in the FF Setup 4, 5, 6 sections.

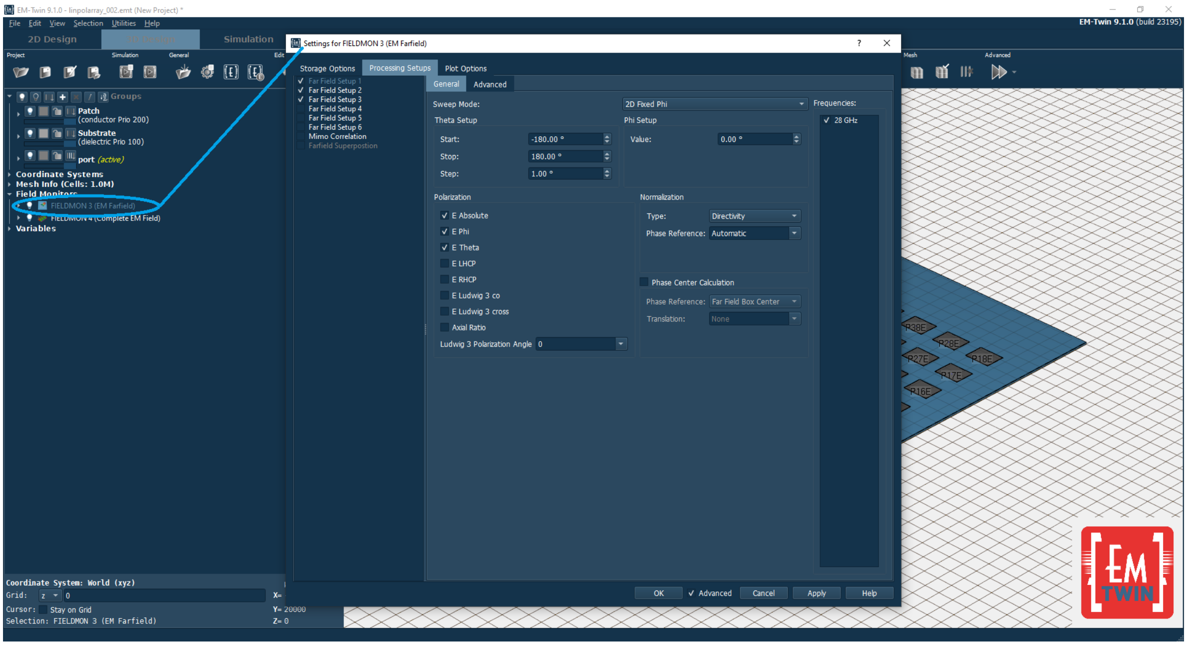


Figure 4.19: Far Field Setup Window

Hint

Far field recording: As a rule of thumb there should be a distance of at least $\lambda_{max}/4$ between radiator and boundary.

Hint

More information about field monitors is available in **EMPIRE-Manual.pdf - Chapter 19: Field Monitors**

5 Simulations and results

After the structure setup, some further simulation parameters, such as frequency range and boundary conditions, can be adjusted in the **Simulation Setup** . To check the discrete structure, the automatic meshing routine can be executed and visualized in a 3D rendered view . By pressing the Simulation button the structure will be checked, discretized and compiled.

When the simulation starts, the evolution of the energy is displayed. As soon as the end criteria is reached, the postprocessing is started for e.g. far field transformation. If field recording boxes have been defined their results can now be visualized by switching to the 3D Result mode and controlled in the Field monitors. These last simulation steps are described in this chapter.

5.1 Set simulation parameters and mesh

When the EM-TWIN™ Wizard is started, some global simulation parameters have already been entered or set by an opened project. To perform the simulation these parameters should be checked.

5.1.1 Set simulation parameters

Simulation parameters which have been entered at the beginning can be edited with the **Simulation Setup** Button .

When using a field source the **Frequency** in **EM Setup** tab is set around the target frequency. The end frequency determines the upper limit of the cell size (e.g. $d_{max} < \lambda_{min}/15$).

If conductor or dielectric objects are present in the structure, they are treated as lossless by default. Other loss treatment can be adjusted here in the **Loss Calculation** section of the **EM Setup** tab.

The **Accuracy** parameters in **EM Setup** tab determine the end criteria as well as conformal meshing (PGA) and pulse modulation (AC only).

The **Mesh Resolution** in **Mesh** tab defines the accuracy of the grid. If it is e.g. set to Coarse (10/3), the number 10 defines at least 10 cells per minimum wavelength and minimum 3 cells are used for the resolution of objects.

In the **Boundary** tab, the boundary conditions and distance can be independently defined for 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 n`: radiating boundary condition with n Perfectly Matched Layers. When the number of layers increases, the reflections produced at the boundary decrease, but the simulation slows down. These matched layers are placed outside the simulation domain.
- `Absorbing n add. space`: radiating boundary condition with n Perfectly Matched Layers, with additional space from the structure.
- `Absorbing sheet`: a sheet with 377 Ω is placed at the boundary which absorbs radiation which is directed normal to the boundary. Used for high directivity radiation or to suppress box mode resonances in simulations.

Hint

More information about simulation parameters is available in **EMPIRE-Manual.pdf - Chapter 23: Simulation Setup**

5.1.2 Set mesh parameters

It is recommended to generate a mesh with the help of the automatic meshing. For this, a few parameters can be adjusted with the **Simulation Setup** button .

The automatic meshing creates a suitable mesh for the entered structure. It can be used at any time e.g. to check the grid of the current structure, and is executed automatically before a simulation is started. The automatic meshing also defines the boundaries of the simulation domain by detecting the objects' extensions and following certain rules coming from far field definition, excitation response, port size, etc. The outermost grid lines define the boundaries of the structure.

In **Resolution**, the mesh accuracy is defined as in Tab. 5.1.

Resolution	Cells per wavelength	Cells for objects
`Very coarse`	8	3
`Coarse`	10	3
`Medium`	15	4
`Fine`	20	5
`Very Fine`	25	6
`Very Fine 2`	28	7
`Very Fine 3`	31	8
`Very Fine 4`	34	9

Table 5.1: Mesh Accuracy

In the **Mode** section, algorithms can be optimized depending on information about the object types (mainly planar or mainly 3-dimensional), different options are:

- `Planar` + `3D` : the grid 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`: grid lines will be preferred for planar structure, i.e. one-third rule grid lines will be favored over generic grid lines from 3D objects.
- `3D`: only 3D objects will be assumed and no optimization for flat metals will be performed.
- `User defined`: only if this mode is activated some parameters of the automatic meshing algorithm in **Automesh Setup Advanced - User Setup** can be adjusted
- `Manual`: this mode is used to switch off the automatic meshing algorithm and to define the grid manually.

In section **Mesh Mode**, it is possible to specify equidistant mesh lines for each axis.

In section or **Mesh Hint** global minimum and maximum cell spacings can be entered.

Hint

More information about meshing is available in **EMPIRE-Manual.pdf - Chapter 22: Meshing and Simulation Domain**

5.2 Simulation

The operation **Start Simulation** covers automatic meshing, detection of the built-in processor and on-line compilation of the structure, solution of the discrete Maxwell's equations until the simulation is finished, and calculation of the frequency domain parameters for the entered frequency range. The status of the simulation is displayed in the **Status** column of the **Simulation** tab.

For advanced users, simulation details can be found in the **Log** window, where compilation, CPU and memory statistics are printed (Fig. 5.1).[1]

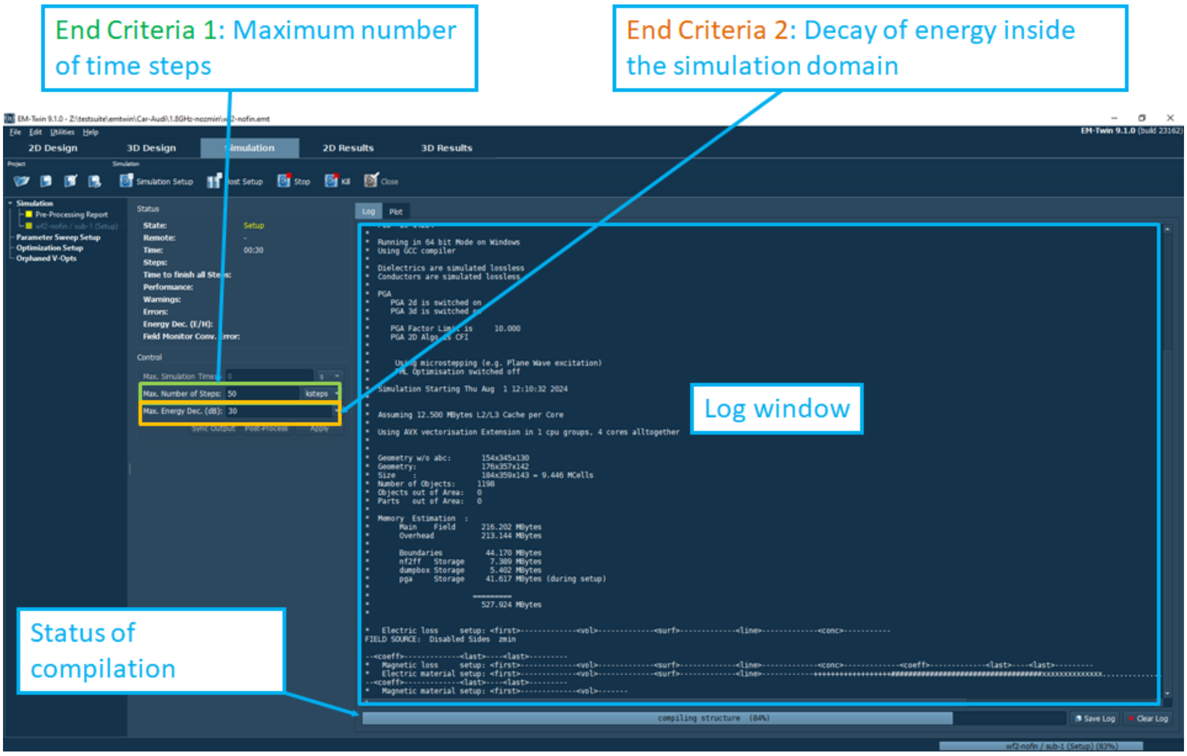


Figure 5.1: Simulation Tab and Log Window

While the simulation is running the evolution of the energy is displayed in time domain in the **Plot** tab.

Hint

Maximum number of time steps and energy decay can be adjusted during the simulation.

[1] After simulation this info is available as fdtd.log in the result folder

Hint

More information about simulaiton is available in **EMPIRE-Manual.pdf - Chapter 24: Simulation**

5.3 Time domain results

After and during the simulation run, time domain results such as voltages, time domain field probes, energy and field monitor convergence can be displayed in the **2D Results** tab (Fig. 5.2). The plot type and the format for the plot can be selected on the left side in the 2D Graph Setup window.

In the file list, the results present in the folder sub-1 are shown in a table according to selected Type. By pressing the **Add Data File** button, the user can load measurement data or results from other sources or simulation projects.

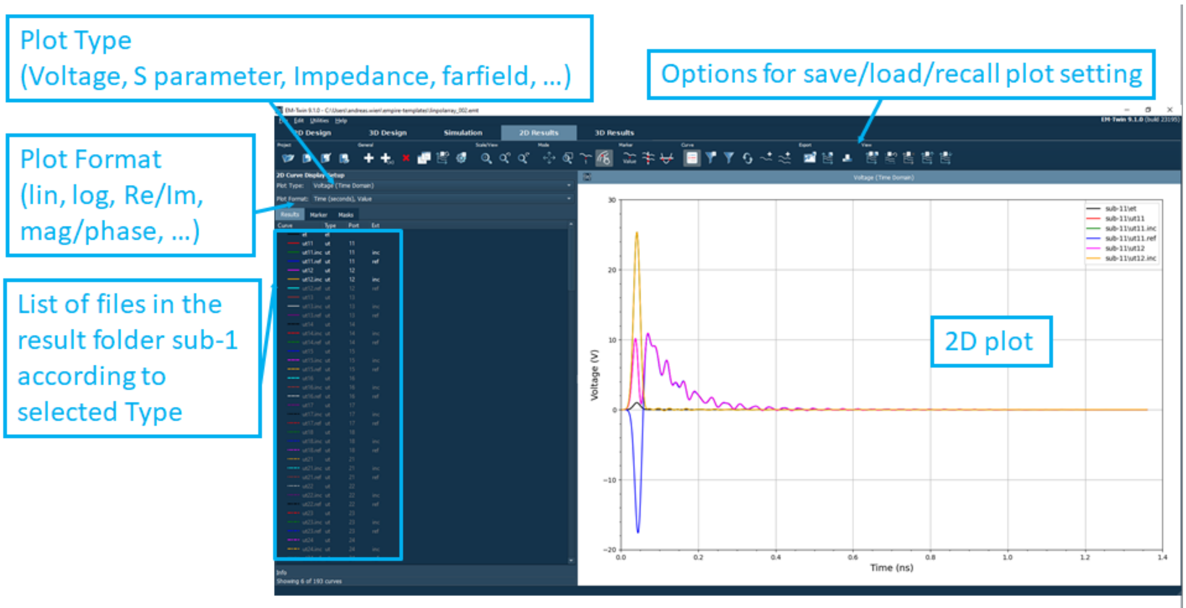


Figure 5.2: Time Domain Graph

5.4 Frequency domain results

After the simulation run and postprocessing, frequency domain results such as field probes, or far field results are available and can be displayed in the **2D Results** tab (Fig. 5.3). Also planar color maps of 3D far field results can be displayed here.

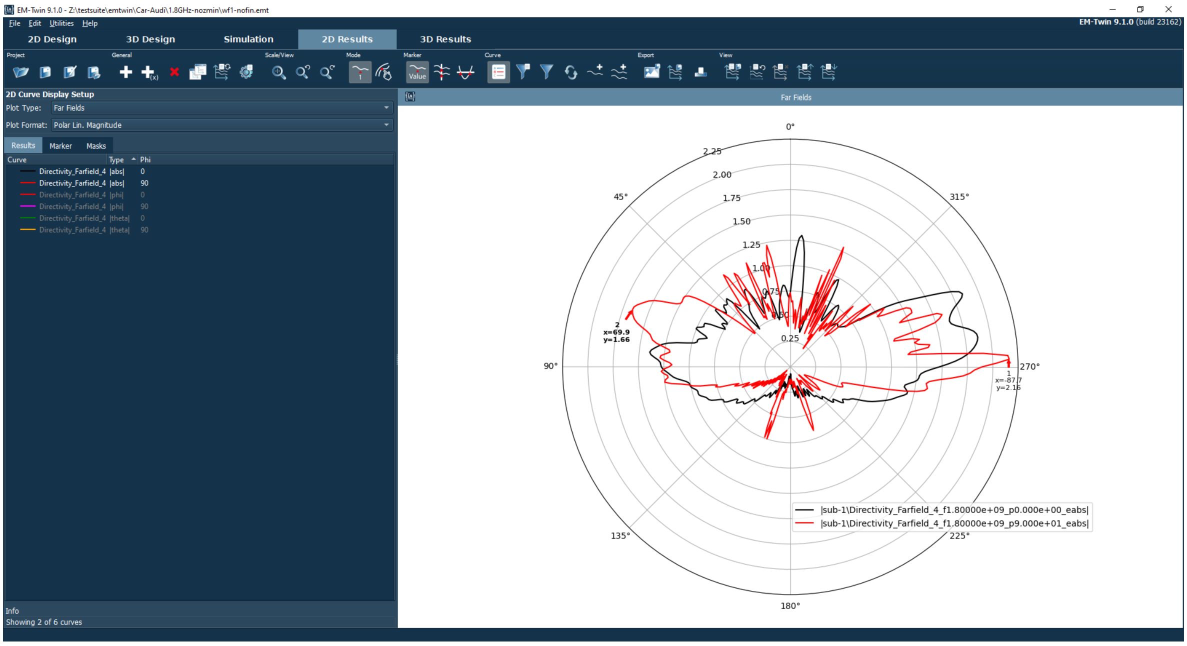


Figure 5.3: 2D Far Field Polar Plot

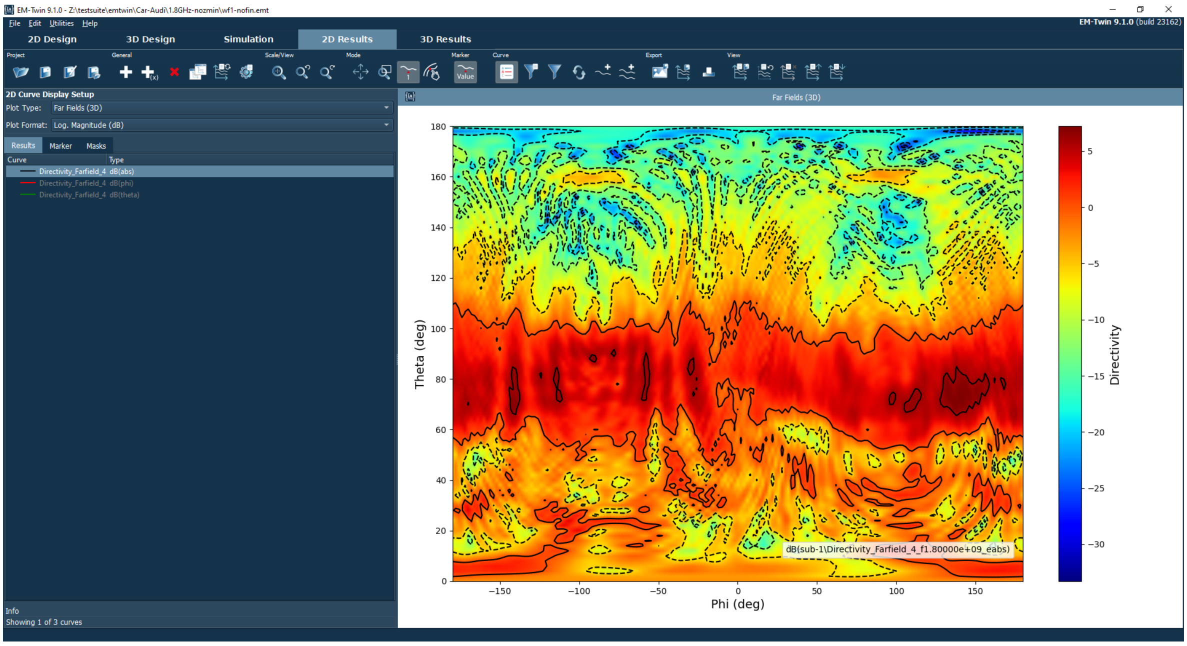


Figure 5.4: 3D Far Field Color Plot

5.5 Near Field Animations

The recorded near fields are shown in the **3D Results** Tab, and the display settings can be defined in the respective field monitor parameters: **3D Plot Option**. Depending on the Field Monitor type, a plane can be selected and set with the help of sliders.

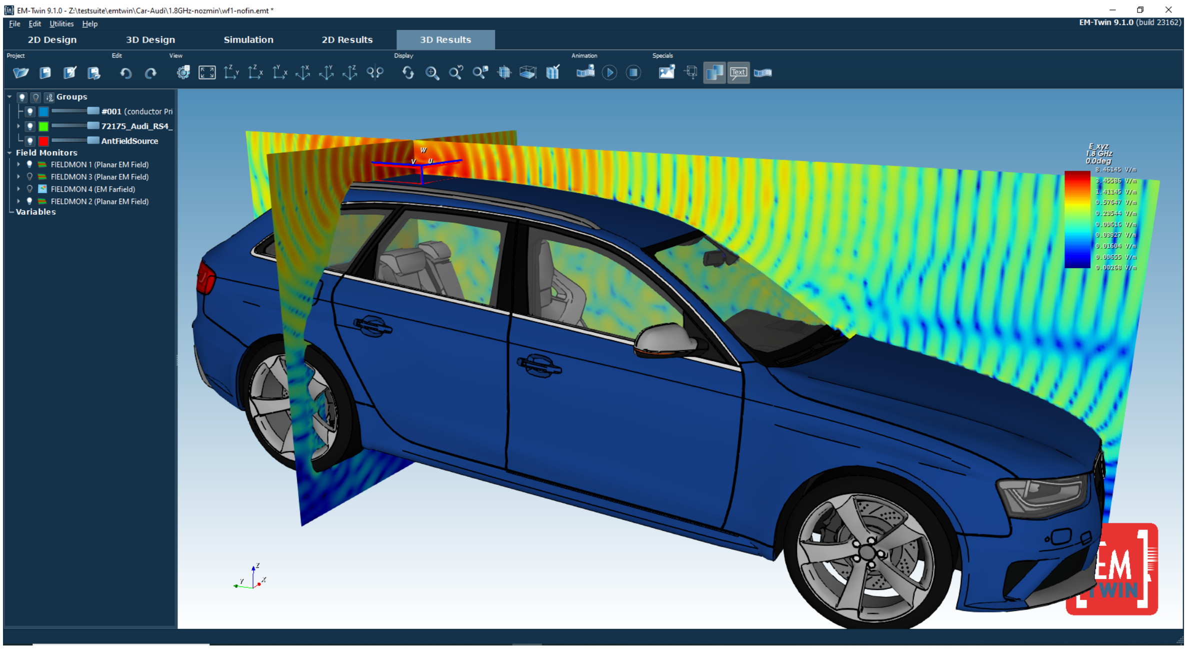


Figure 5.5: Near Field 3D Plot

5.6 Far Field Animations

The recorded 3D far fields are shown in **3D Results** Tab (Fig. 5.6), and the far field display settings can be defined in the respective field monitor parameters: **3D Plot Option**.

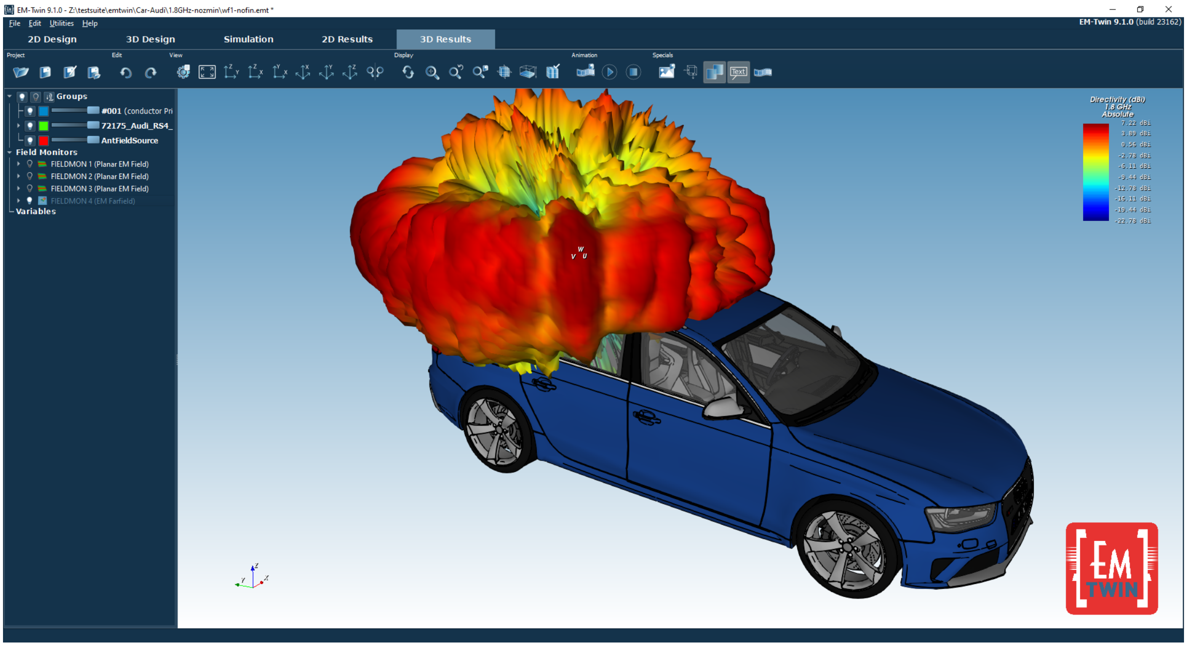


Figure 5.6: Far Field 3D Plot

If far field patterns have been calculated, they will be detected in the Filename box, and can be selected for their display. Plot style and legend keys setup options are similar to the near field animation options.

Hint

More information about results is available in **EMPIRE-Manual.pdf - Chapter 25: Results**

6 Finding Further Information

Manual

A more detailed and comprehensive description of the EM-TWIN™ software is available as EMPIRE-Manual.pdf on media in `/manual/`, installation tree or Help menu.

Tutorials

In order to learn the software efficiently, several step-by-step tutorials are available at program start.

Email Support

During the evaluation period the user can send specific questions directly to

em-twin.support@imst.de.

The EM-TWIN™ Support Team is backed up with very experienced RF designers, programmers and high educated EM modelers. Help is available usually within 24 hours on German working days. Local support teams are thoroughly trained by the EM-TWIN™ team and are available for support without time shift. In case of modeling problems the user is requested to send the Version and Build number and the following files, so the problem can be traced:

- `.emt`: EM-TWIN™ input file
- `.surf.dat`: EM-TWIN™ field source file
- `.log`: Log files located in root and sub folders