Agilent Technologies

EMPro 2010
May 2010
EMPro Quick Start

Acknowledgments
Mentor Graphics is a trademark of Mentor Graphics Corporation in the U.S. and other countries. Microsoft®, Windows®, MS Windows®, Windows NT®, and MS-DOS® are U.S. registered trademarks of Microsoft Corporation. Pentium® is a U.S. registered trademark of Intel Corporation. PostScript® and Acrobat® are trademarks of Adobe Systems Incorporated. UNIX® is a registered trademark of the Open Group. Java™ is a U.S. trademark of Sun Microsystems, Inc. SystemC® is a registered trademark of Open SystemC Initiative, Inc. in the United States and other countries and is used with permission. MATLAB® is a U.S. registered trademark of The Math Works, Inc.. HiSIM2 source code, and all copyrights, trade secrets or other intellectual property rights in and to the source code in its entirety, is owned by Hiroshima University and STARC.

The following third-party libraries are used by the NlogN Momentum solver:

"This program includes Metis 4.0, Copyright © 1998, Regents of the University of Minnesota", http://www.cs.umn.edu/~metis , METIS was written by George Karypis (karypis@cs.umn.edu).

Intel@ Math Kernel Library, http://www.intel.com/software/products/mkl

SuperLU_MT version 2.0 - Copyright © 2003, The Regents of the University of California, through Lawrence Berkeley National Laboratory (subject to receipt of any required approvals from U.S. Dept. of Energy). All rights reserved. SuperLU Disclaimer: THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

AMD Version 2.2 - AMD Notice: The AMD code was modified. Used by permission. AMD copyright: AMD Version 2.2, Copyright © 2007 by Timothy A. Davis, Patrick R. Amestoy, and Iain S. Duff. All Rights Reserved. AMD License: Your use or distribution of AMD or any modified version of AMD implies that you agree to this License. This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser

Errata The ADS product may contain references to "HP" or "HPEESOF" such as in file names and directory names. The business entity formerly known as "HP EEsof" is now part of Agilent Technologies and is known as "Agilent EEsof". To avoid broken functionality and to maintain backward compatibility for our customers, we did not change all the names and labels that contain "HP" or "HPEESOF" references.

EMPro Quick Installation

Download the EMPro installation files from the EMPro download page .

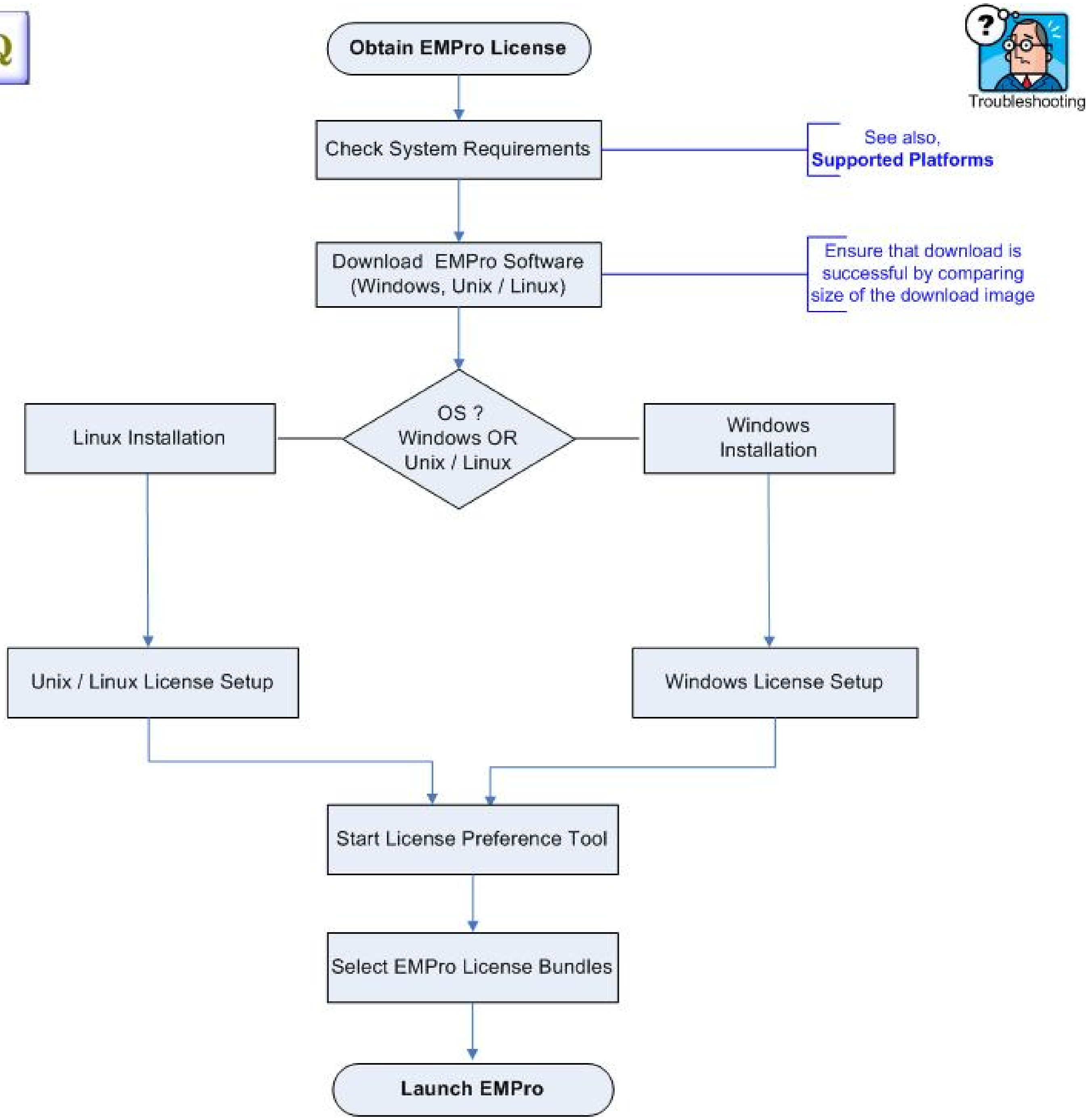
EMPro can be installed on a PC running Windows and Linux operating systems.

The following flowchart illustrates quick installation steps that will help you to install EMPro quickly and create a complete license setup.

ℹ	Click on the respective box in the flowchart below for further details on the selected topic.

EMPro Installation Flowchart

FAQ



Obtaining EMPro License

You can place EMPro license request for any of the following requirements:

- New Sale License Request
- New Release License Request
- Renew Expired License Early Access
- License Request Change / Transfer
- Evaluation License / Token Redemption / Renewal Request

Click here to place your EMPro License request.

Back to Installation Flowchart

Supported Platforms

EMPro supports the following Windows and Linux platforms:

Operating System	32-bit	64-bit
Windows/VISTA/Windows 7	Microsoft Windows XP Professional SP3 Microsoft Windows Vista Enterprise SP1	Microsoft Windows XP Professional x64 Microsoft Windows Vista Enterprise SP1 64-bit edition
LINUX	Redhat RHEL WS 4.x Novell SUSE SLES 10	Redhat RHEL WS 4.0 Novell SUSE SLES 10 (64-bit AMD Opteron and Intel EM64T processors)

Back to Installation Flowchart   Troubleshoot - Supported Platforms

System Requirements

The system requirements for EMPro 2010 include:

- 1 GB RAM (minimum), 2 GB or more (recommended)
- 1280 x 800 display resolution
- ATI Radeon 7500 or NVidia GeForce 4 or a newer video card
- 3 GB or more hard disk space for complete installation of software and all the example files

Back to Installation Flowchart

Download EMPro Software

Click here to download EMPro installation file of Windows/Vista and Linux.

You can also use Agilent Download Manager to download EMPro installation file.

> ⚠ Make sure your download is successful by comparing the downloaded file size with the one in download page. In case of any difference please download the file again.

Back to Installation Flowchart   Troubleshoot - EMPro Download

Installation

Windows Quick Installation

Note
Use this condensed installation procedure if you are experienced in the installation of Agilent EEsof products. If you have installed an *Early Access* version of EMPro, you should uninstall it before installing this version.

To install EMPro on a Windows PC:

1. After downloading the EMPro windows installation image from the EMPro download page , unzip the file contents to the local hard disk.
2. Navigate to the directory where you saved the downloaded image file and unzip it.
3. Exit all Windows programs and run setup.exe from the extracted files.
4. When the installation wizard appears, follow the on screen instructions to start the installation. When the installation is complete, click **Finish**.
    - Install your Licenses. For instructions on installing the Licenses, refer to *Installing Your Licenses* (license).
    - Install the *Acresso FLEXid* software-security hardware key to your PC's parallel port, or use your PC LAN card's Ethernet ID. For instructions and more information, refer to *Installing Your Licenses* (license).
5. Launch EMPro. Do this by selecting **EMPro** > **EMPro 2010** > **EMPro 2010 (32-bit GUI)** (for 32-bit systems, replace 32 by 64 on 64-bit systems) from your *Start* menu. If you are using license bundles, select a bundle using the *Agilent License Preference Tool* prior to running EMPro. This tool is described in *Licensing EMPro* (license) in the EMPro documentation. The licensing tool is available from the start menu by selecting **EMPro** > **EMPro 2010** > **EMPro Tools** > **License Preference Tool**.

Note
Before launching EMPro, click here for *EMPro License Setup* (license).

Back to Installation Flowchart   Troubleshoot - Installation

Linux Quick Installation

Note
Use this quick installation procedure if you are experienced in installing Agilent EEsof products. If you have installed an *Early Access* version of EMPro, you should uninstall it before installing this version.

To install EMPro on a Linux PC:

1. Log on to the system where you want to install EMPro.
2. Download and untar the EMPro installation image from the EMPro download page .
3. Change directories to the directory where the extracted tar files are located.
4. To start the *Setup* program use the following command:
   `./SETUP.SH`
5. When the EMPro 2010 installation window appears, you can begin the installation. Details about each window are available in *Detailed Installation* (install). When the installation is complete, note the *License ID* (hostid) then click **Done** to exit the program.
6. Use the *FLEXnet* security licenses supplied by Agilent EEsof to set up a *license.lic* file. For more information, refer to *Installing Your Licenses* (license).

7. Place the *license.lic* file in the *licenses* sub-directory of your EMPro installation directory and start FLEXnet. For more information, refer to *Licensing EMPro* (license).
8. If you are using license bundles, select a bundle using the *Agilent License Preference Tool* prior to running EMPro . This tool is described in *Using the Agilent License Preference Tool* (license).
9. Launch EMPro. To do this, navigate to the directory in which EMPro is installed and select ***bin/Linux-i686RHEL4*** (for 32-bit systems) or ***bin/Linux-x86_64*** (for 64-bit systems) and use the command:

   ./startempro --driver=x11

Windows License setup

To setup the licenses for Windows Installation, visit *Windows License Setup* (license).

Back to Installation Flowchart

Linux License setup

To setup the licenses for Linux Installation, see *Linux License Setup* (license).

Back to Installation Flowchart

Setting the Display

If you want to run EMPro from a remote computer and you want the display to appear on your local machine, you will need to set the DISPLAY environment variable:

```
setenv DISPLAY : 0.0 (C-Shell)
DISPLAY = : 0.0 (Korn Shell, Bourne Shell)
export DISPLAY
```

For a Sun Ray file server and diskless terminals using Solaris 8, you will need to set the *DISPLAY* environment variable:

```
setenv DISPLAY $Display (C-Shell)
set DISPLAY = $Display (Korn Shell, Bourne Shell)
export DISPLAY
```

For details on using the Sun Ray appliance, refer to the Sun Microsystem website at: http://wwws.sun.com/sunray/index.html

Back to Installation Flowchart   Troubleshoot - Installation

Launch EMPro

Launching EMPro in Windows

Your FLEXnet license file must be properly configured and installed before you can run EMPro. To set up your license file, follow the instructions in *Windows License Setup* (license).

To run EMPro from the Start menu, select **Programs** > **EMPro** > **EMPro 2010** > **EMPro 2010 (32/64 bit GUI)**.

The choices available are:

- **EMPro Documentation:** Brings up your Web browser and the starting point for accessing EMPro documentation. The documentation files are accessed from the location in which they are installed (if you chose to install documentation).
- **Launch EMPro in Windows:** Launches the EMPro Main window, enables the 32-bit simulators, and the use of the various EMPro Suites, features and modules you have licensed. (If EMPro is installed on a 64-bit operating system and you want to use the 64-bit simulator, choose EMPro (64-bit GUI).
  If you are not familiar with EMPro, choose *Help* > *Topics and Index* > *Quick Start* for help on getting started with EMPro.
- **Uninstall EMPro 2010:** Launches the Uninstall Program.

Launching EMPro in Linux

Environment variables must be set before you can run EMPro.Your FLEXnet license file must be properly configured and installed before you can run EMPro. To set up your license file, follow the instructions in *Linux License Setup* (license).

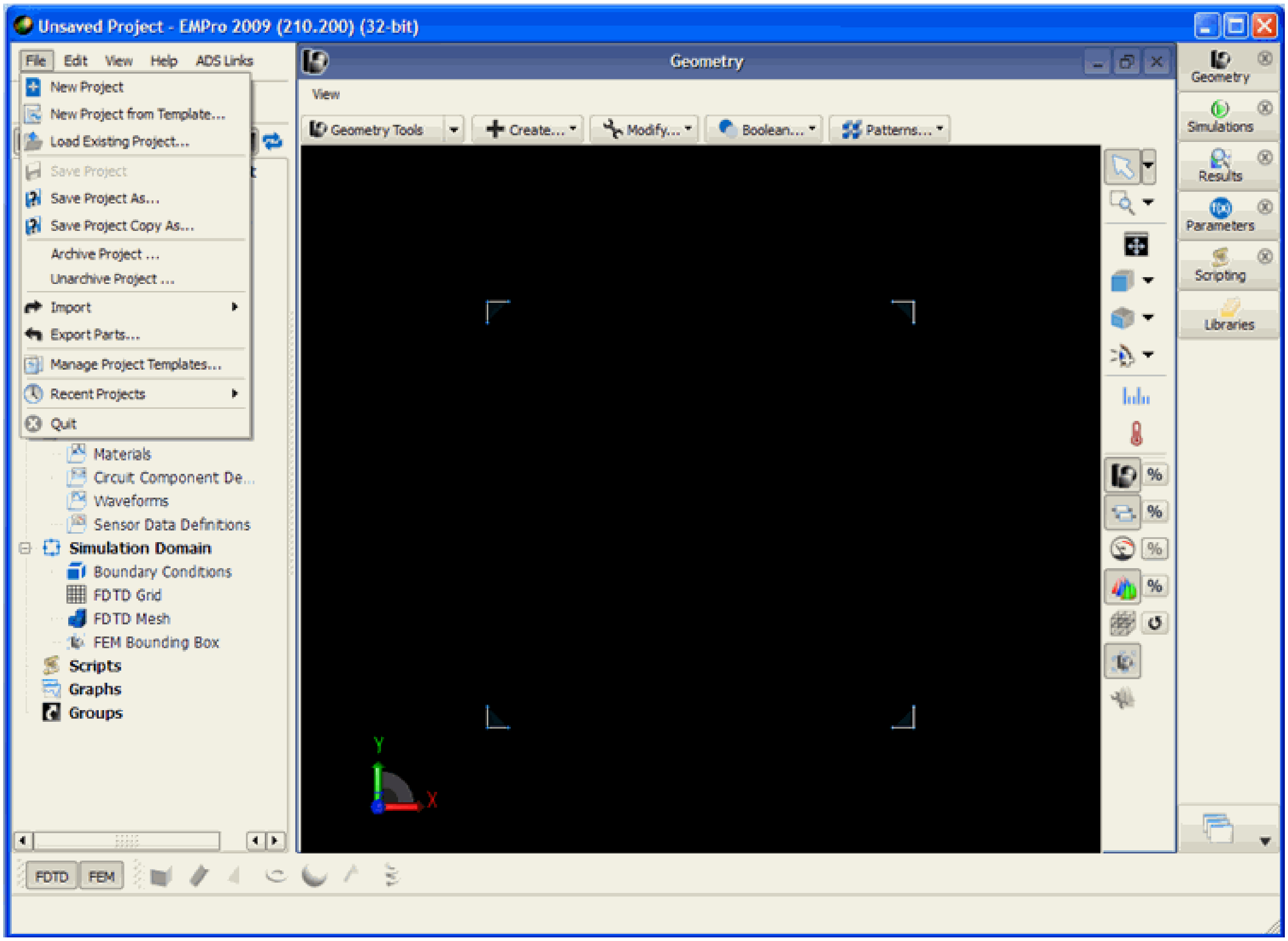
Choose **Help** > **Topics and Index** > **Quick Start** for help on getting started with EMPro.

Managing EMPro Projects

EMPro uses projects to organize and store the data generated when you create, simulate, and analyze designs to accomplish your design goals.

A project includes circuit, layout, simulation, analysis, and output information on the designs that you create, along with any links you add to other designs and projects.

Use the **File** menu in the EMPro Main window to create and open projects.



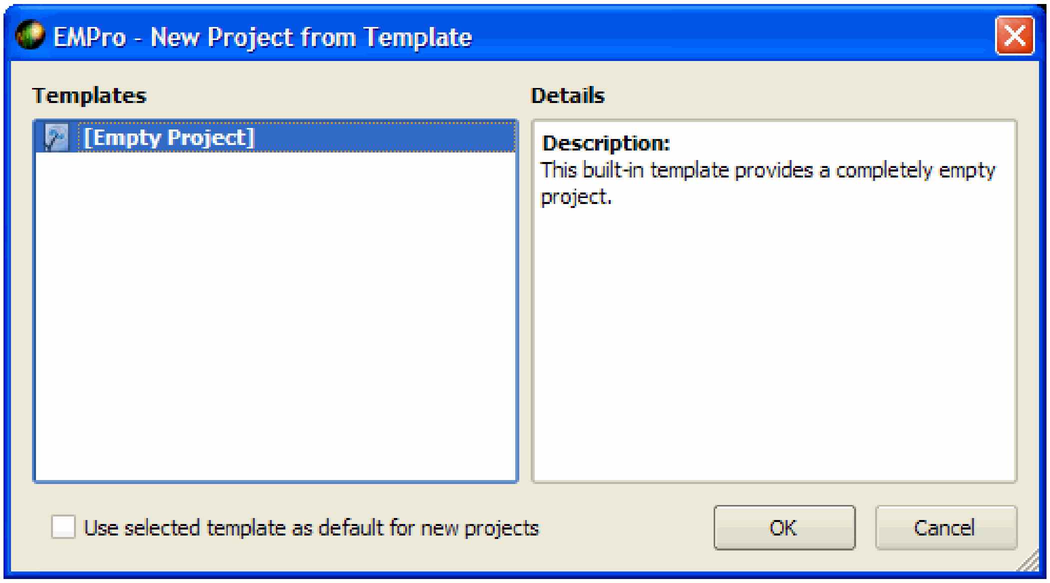
Creating Projects

Use the *Main* window to create a project and organize your designs.

Choose **File > New Project** to create a new project in EMPro.

Creating Projects from a Template

1. Choose **File > New Project From Template** to open the following dialog box:



2. Click **OK** to open the project.

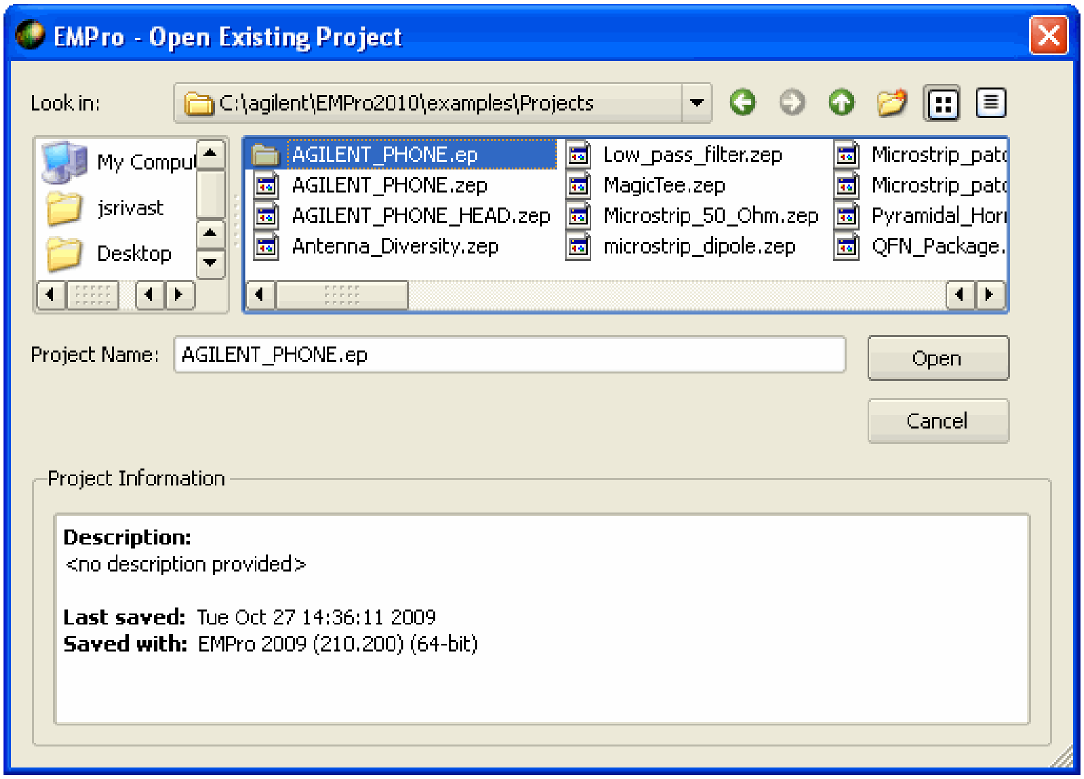
Opening Existing Projects

Use the *Main* window to reuse and load the existing projects without the requirement to include all the individual parts manually to make a project.

> **Note**
> You can open only one project at a time.

When you begin to open a project, you are prompted to save any changes you have made in the currently open project before it is closed automatically.

1. Choose **File** > **Open Existing Projects** to open the following dialog box:

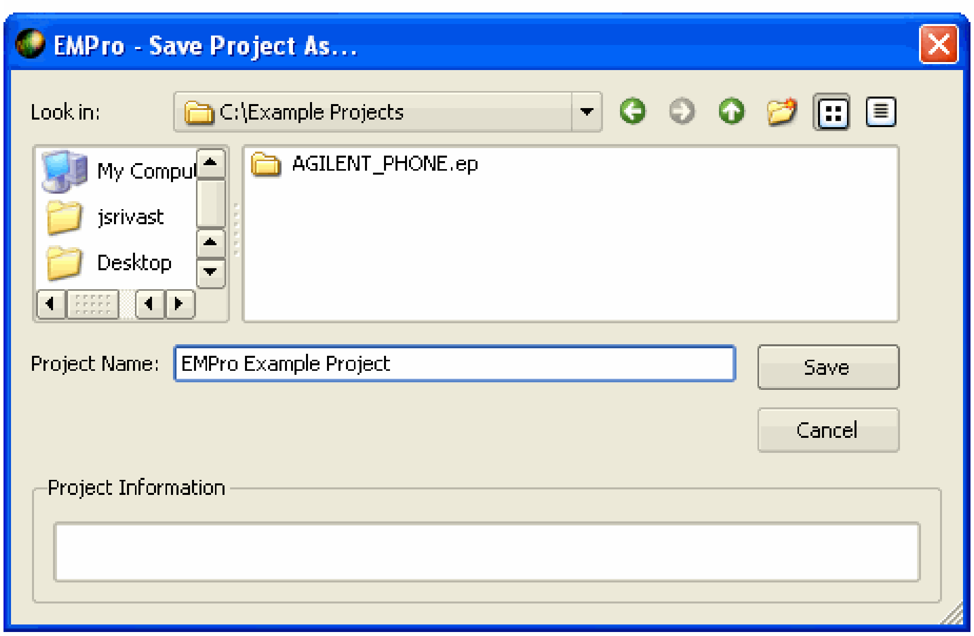


2. Select the project you wish to open and click **Open** to open the project.

Saving a Project

To save a new project to a specified directory:

1. Choose **File** > **Save Project As** to save the project. The following dialog box is displayed:

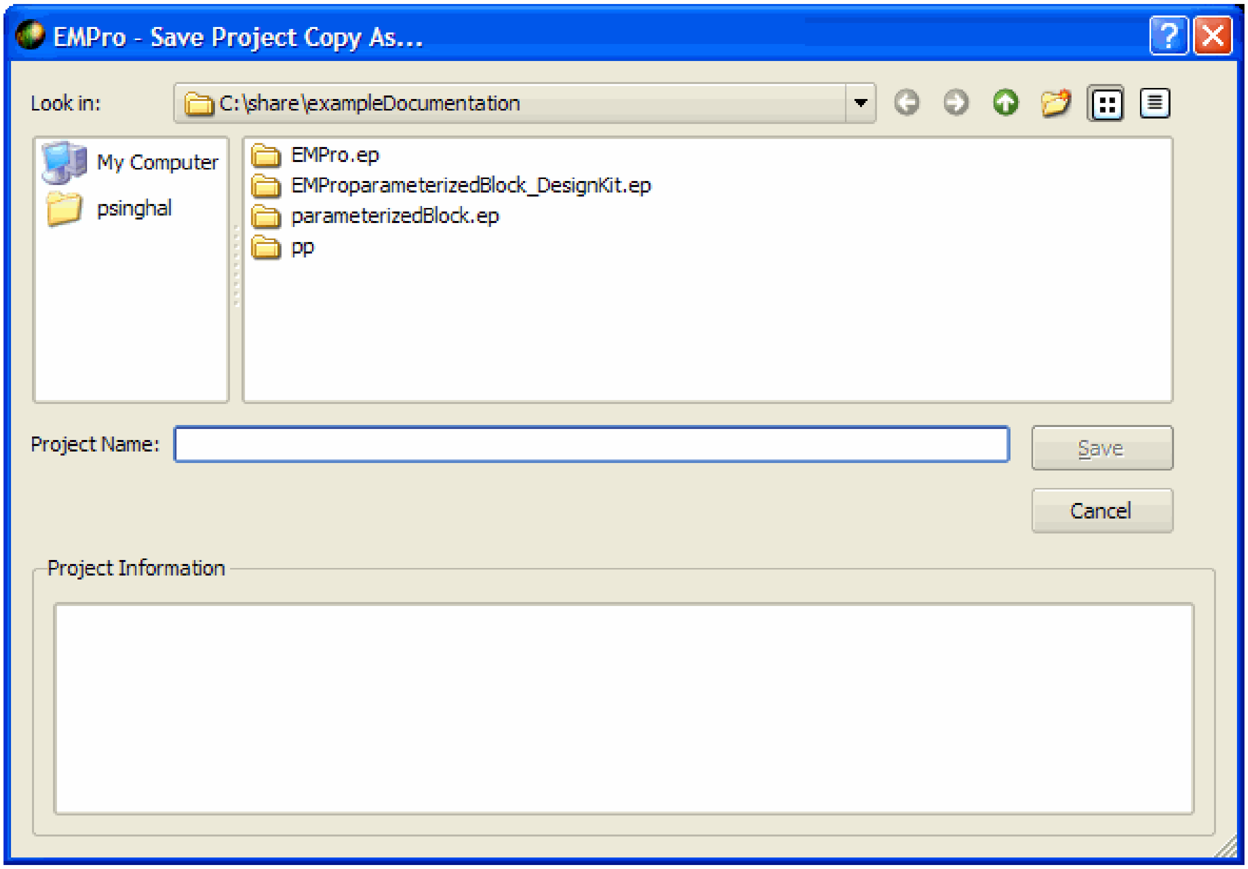


2. Type the name of the project and click **Save** to save the project.

Saving a Copy of the Project

Copying a project directory and its contents to a new project directory allows you to save time and effort by using an existing project as a template.

1. Choose **File** > **Save Project Copy As** to save the project. The following dialog box is displayed:

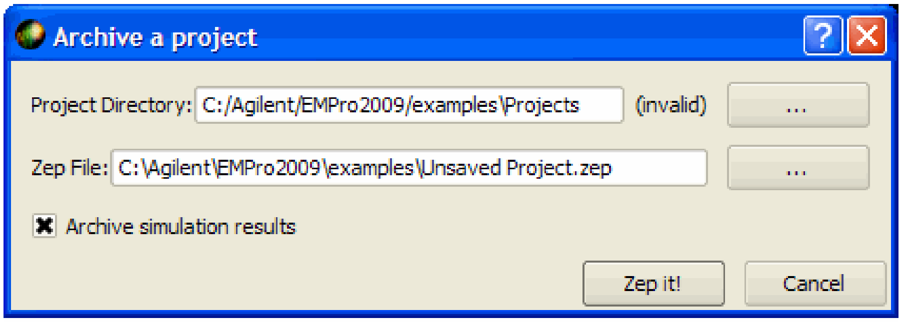


2. Type the name of the project and click **Save** to save the project.

Archiving Projects

Archive/Unarchive projects to transfer a compact project archive. Creating a single file for a project simplifies transferring projects to another file system or to another location on the same file system.

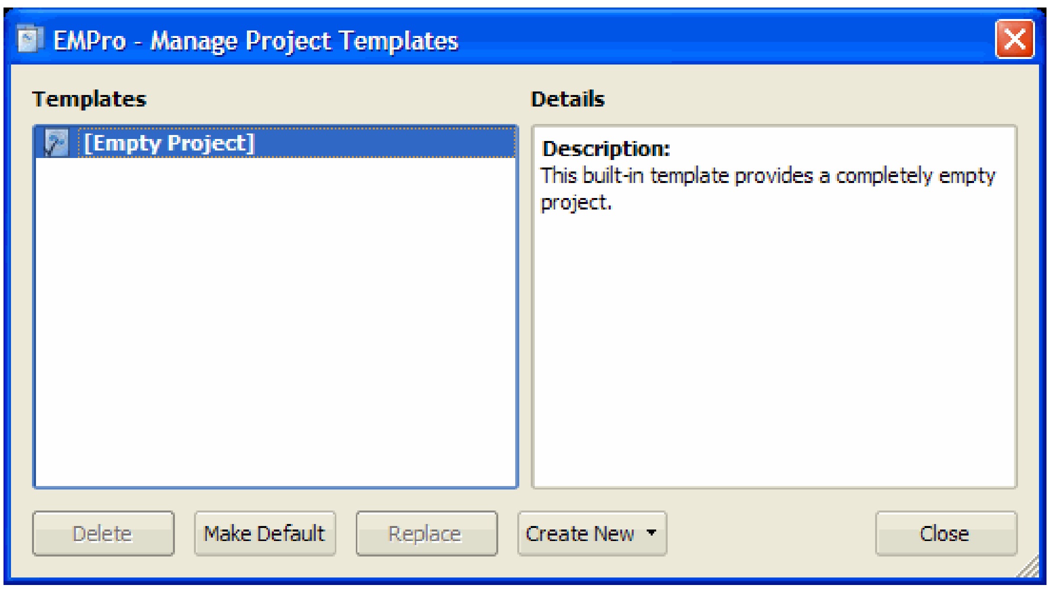
Choose **File > Archive Project** and use the *Archive a project* dialog box to locate and archive the project.



Managing Project Templates

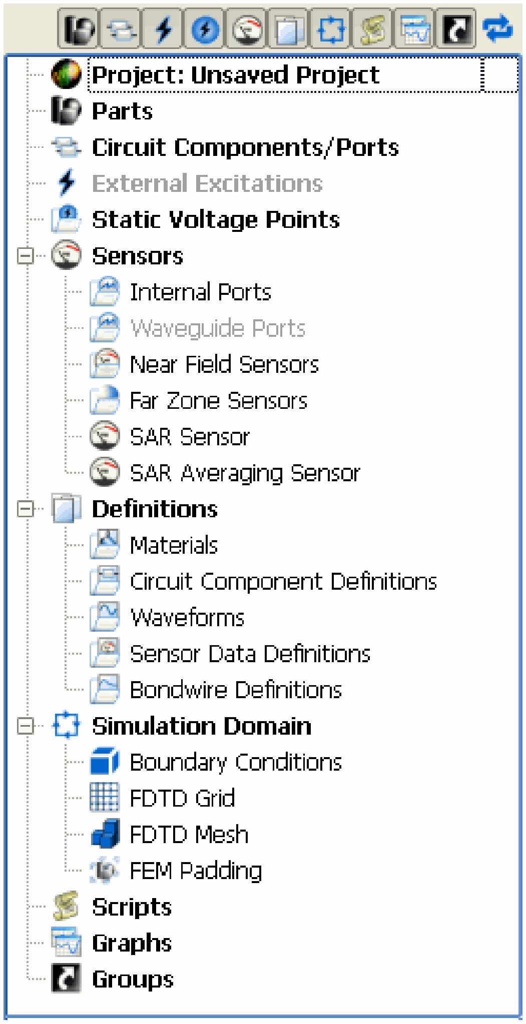
EMPro allows you to create a new template or to assign a default template to be loaded whenever a new project is opened. You can delete, replace, and modify projects, as well as create a new template from the current project.

Choose **File > Manage Project Templates** and use the following dialog box to manage a project.



Using the Project Tree

The EMPro Project Tree provides a tree-structured representation of the active project, as shown in the following figure:



It is organized into the following branches:

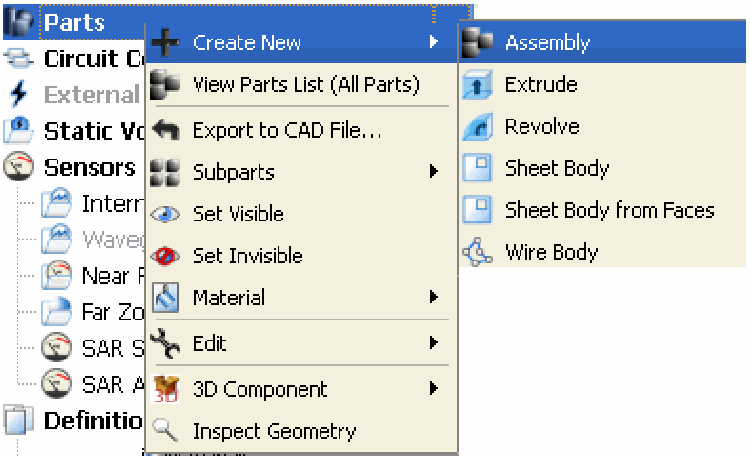
- Parts
- Circuit Components/Ports
- External Excitations
- Static Voltage Points
- Sensors
- Definitions
- Simulation Domain
- Scripts
- Graphs
- Groups

The EMPro Project Tree is easy to manipulate by means of branch and object toggle buttons.

Branches of the Project Tree

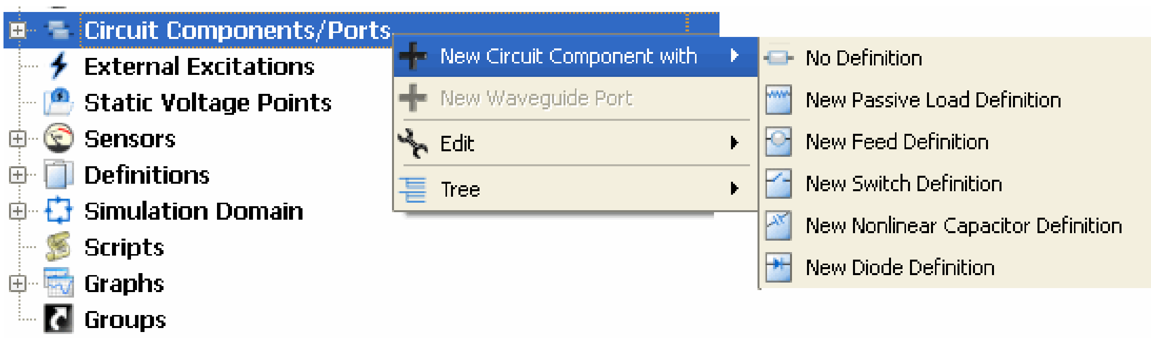
Parts

The *Parts* branch organizes the physical parts of a project. It also lists material definitions and modeling operations applied to any parts object in the tree.
It is possible to organize similar parts objects in groups with an Assembly by right-clicking and selecting **Create New: Assembly**, as shown in the below illustration.



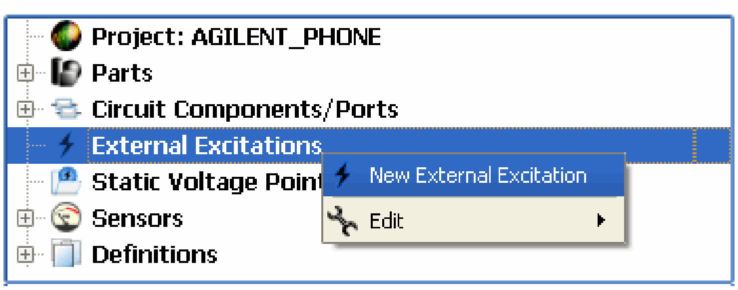
Circuit Components/Ports

The *Circuit Components/Ports* branch organizes discrete circuit components in a project.



External Excitations

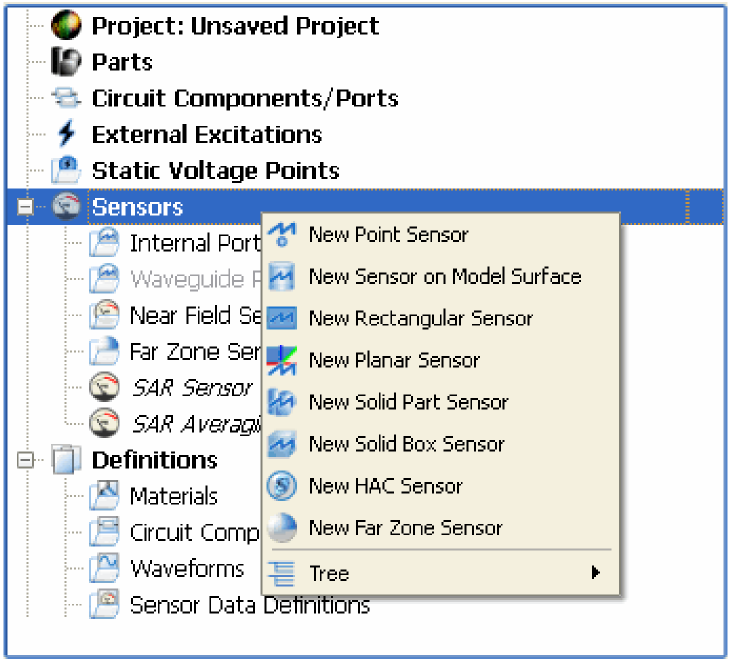
The *External Excitations* branch organizes the external excitations applied to a project.



Sensors

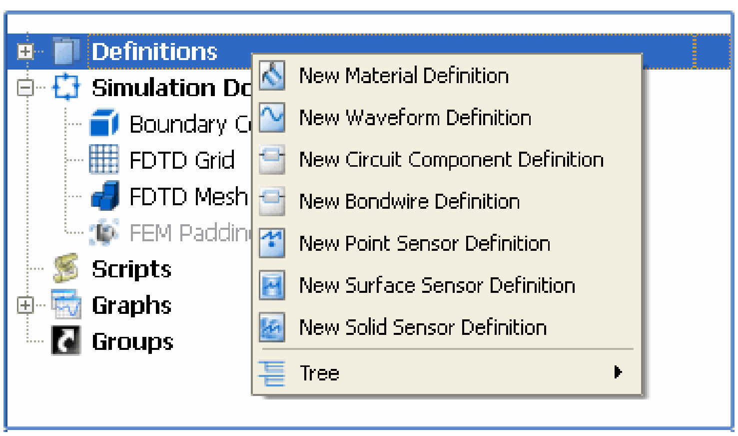
The *Sensors* branch organizes the sensors defined in a project. Sensors are responsible for

saving the data collected during a calculation.
They are added by right-clicking on Sensors branch of the Project Tree and choosing the required sensor.



Definitions

The *Definitions* branch stores definitions that can be applied to or shared with other objects within the project.
To add a new definition object, right-click on the Definitions branch and select the desired definition.

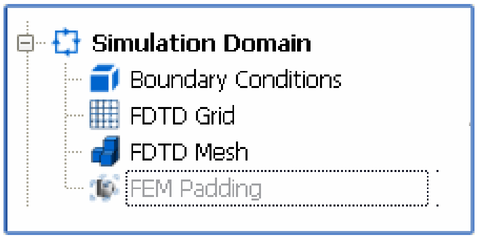


Simulation Domain

The *Simulation Domain* branch stores definitions associated with the outer boundaries of the project, as well as the grid and mesh. It also includes information about the FEM Padding Editor.

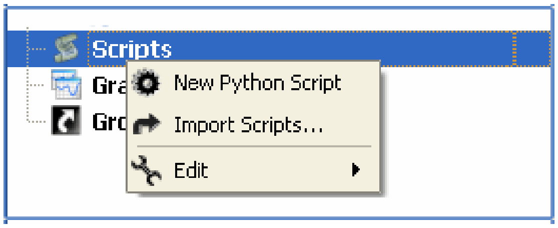
- Double-clicking the **Boundary Conditions** icon will bring up the Boundary Conditions Editor.
- Double-clicking the **FDTD Grid** icon will bring up the FDTD Grid Tools dialog box,

used to specify the characteristics of the grid.
- Double-clicking the **FDTD Mesh** icon will enable Mesh View.
- Double-clicking the **FEM Padding** icon will bring up the FEM Padding Editor.



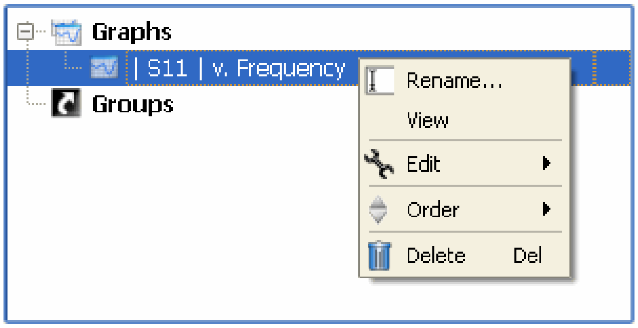
Scripts

The *Scripts* branch stores user-defined scripts.
Right-click this branch to add a new script or to import an existing macro or function script to the project.
You can execute or edit the script in the Scripting workspace window.



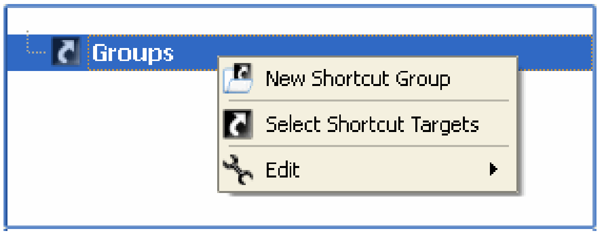
Graphs

The *Graphs* branch organizes the graphical output associated with data collected during a calculation



Groups

The *Groups* branch allows you to create fully customizable short-cut groups that may include any grouping of objects (for example, Parts objects, Sensor objects, Definition objects, etc.).


Groups
New Shortcut Group
Select Shortcut Targets
Edit

Using Workspace Windows

EMPro consists of workspace windows that are a series of windows with each having its own designated function.

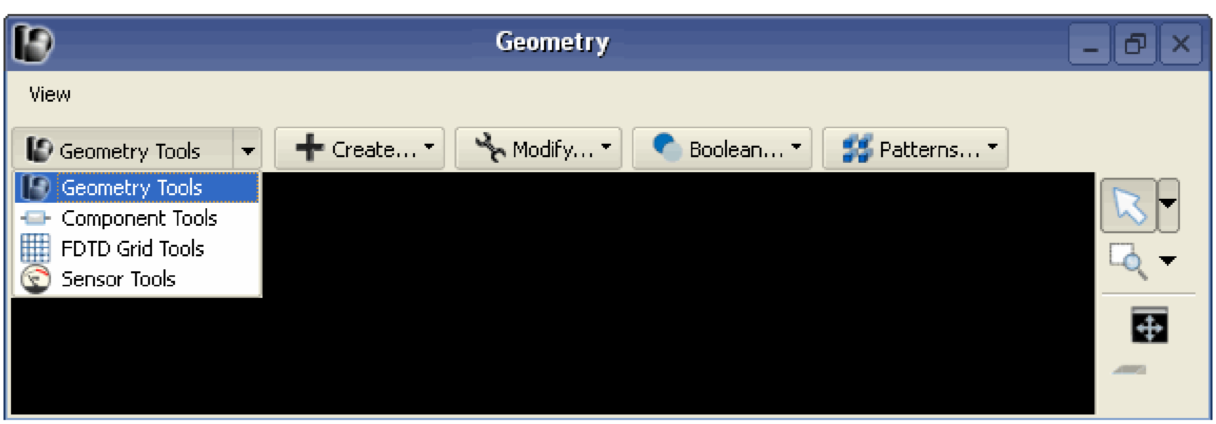
You can use the following workspace windows in EMPro:

- Geometry
- Simulations
- Results
- Parameters
- Scripting
- Libraries

The following figure highlights the workspace windows:

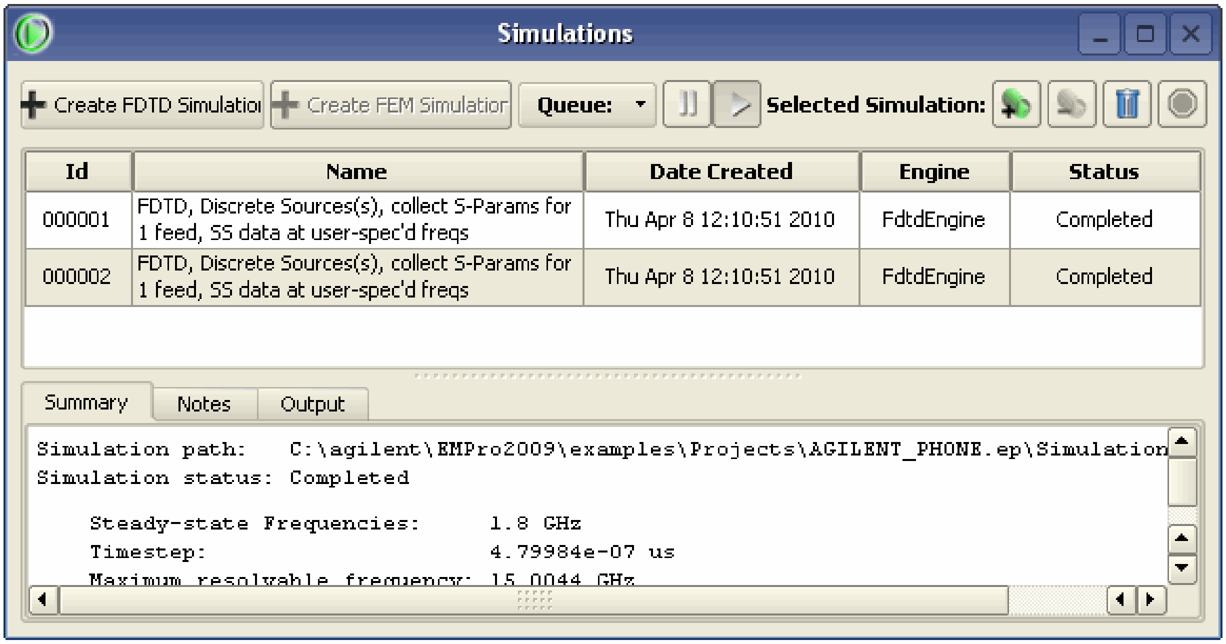
Geometry

The *Geometry* workspace window comprises the main project viewing area.
The window contains four main tools used to add and edit the fundamental elements of a project: Geometry Tools, Component Tools, FDTD Grid Tools, and Sensor Tools.



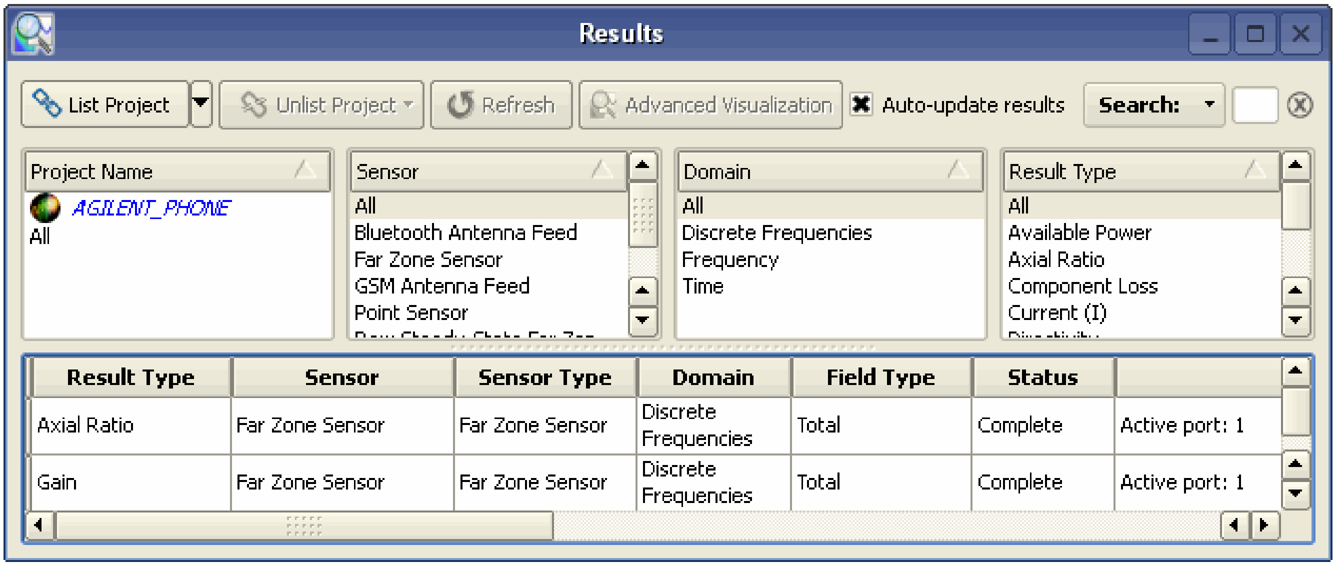
Simulations

The *Simulations* workspace window provides the main interface to define simulations to send to the calculation engine.
This workspace window stores definitions such as source types, parameter sweeps, S-parameters, frequencies of interest.



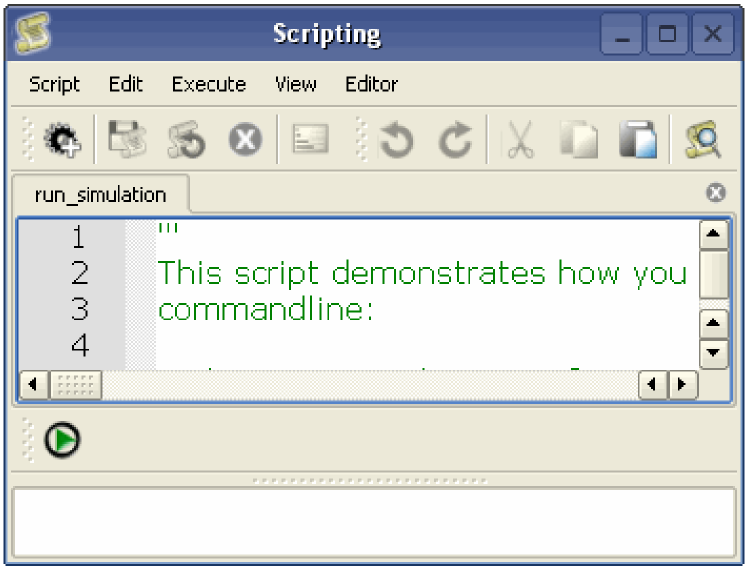
Results

The *Results* workspace window stores all of the results available for a particular project. It is also possible to load the results of a past project into this window without having to load the entire project itself.



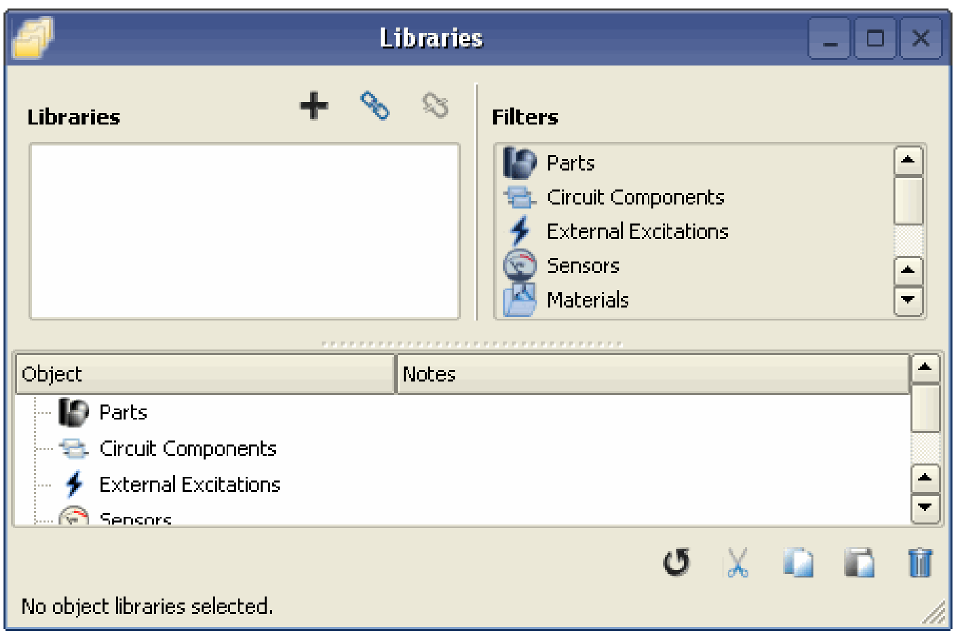
Scripting

The *Scripting* workspace window allows you to view, edit and execute the scripts.



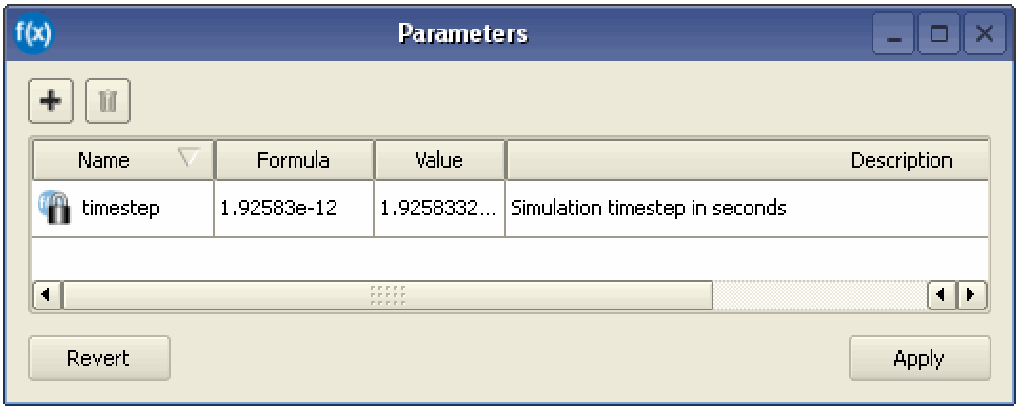
Libraries

The *Libraries* workspace window allows you to customize the library for materials, geometries, etc... for their repeated usage. It allows you to add or access customized libraries.



Parameters

The *Parameters* workspace window enables you to create, edit, and delete parameters that are referenced.


Parameters
Name
Formula
Value
Description
timestep
1.92583e-12
1.9258332...
Simulation timestep in seconds
Revert
Apply

Using the Geometry Workspace

In this section, you will learn how to:

- Create, edit, and import the geometry of your EMPro project
- Align your geometry and view it from any angle

This section focuses on geometric modeling within the EMPro interface. It begins by describing the basic functions available within the *Geometry* workspace window, where the project geometry is created.

The *View Tools* option which is available for orienting the perspective of the simulation space, is detailed in the following section.

Under *Geometry Tools*, you will be introduced to the 2-D and 3-D modeling tools, modification and boolean operations, and patterned arrays that are available within this dialog. In addition to creating geometry from scratch within Geometry Tools, external files, such as CAD and voxel files, can also be imported and modified.

After the geometry is created or imported into a project, it is often necessary to adjust an object's orientation. EMPro has the capability of orienting not only geometric parts, but also other physical parts, such as components and sensors.

The final section details the *Specify Orientation* tab that is available during any editing session involving a physical part.

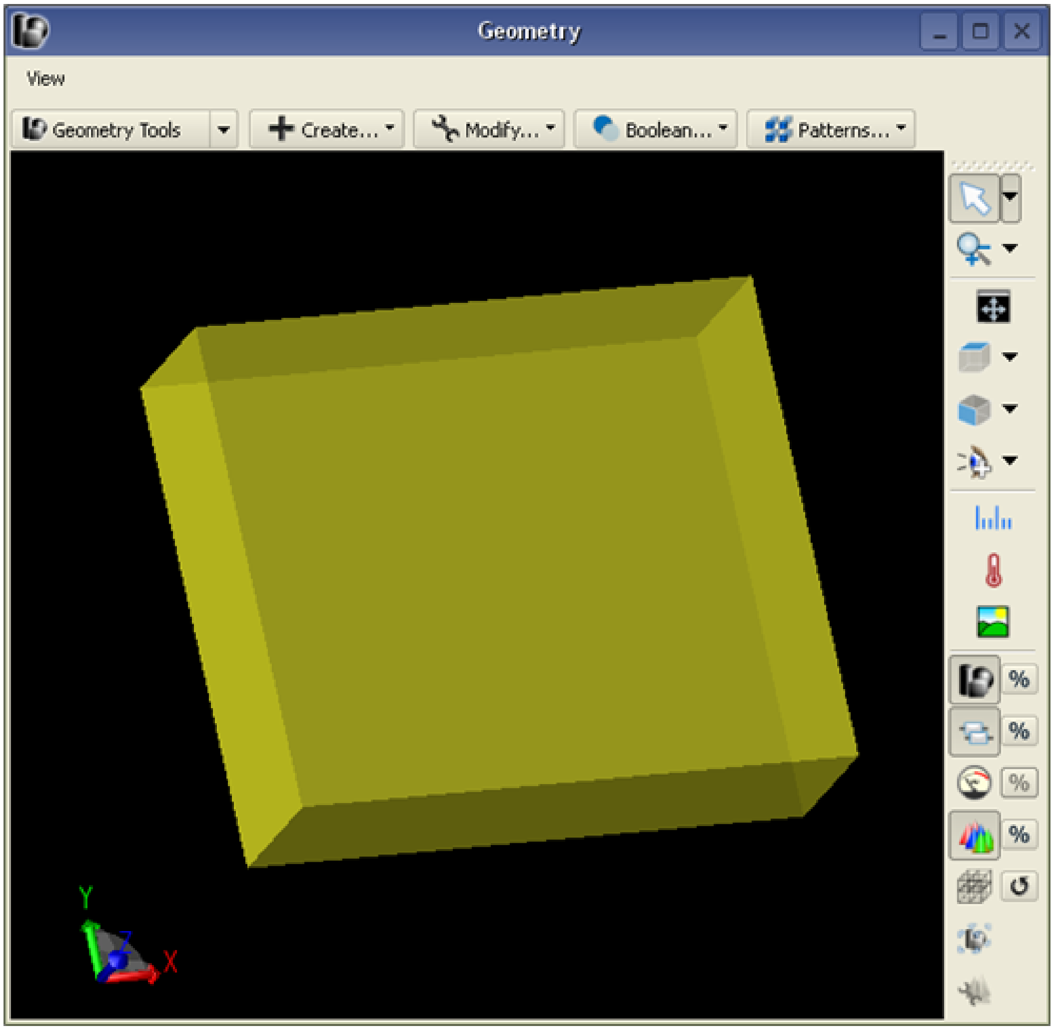
Geometry Workspace Window

The *Geometry* workspace window provides the graphical interface of the EMPro project. Within the window, there are a series of *View Tools* along the right of the viewing area that can be used to manipulate the view of the simulation space at any time. Along the top of the viewing area, the window contains *Geometry Tools* used to create and edit various aspects of the project geometry, in preparation for the final calculation.

The first drop-down menu in the upper-left part of the *Geometry* workspace window contains four different tools:

- Geometry Tools
- Component Tools
- FDTD Grid Tools
- Sensor Tools

The Geometry workspace window



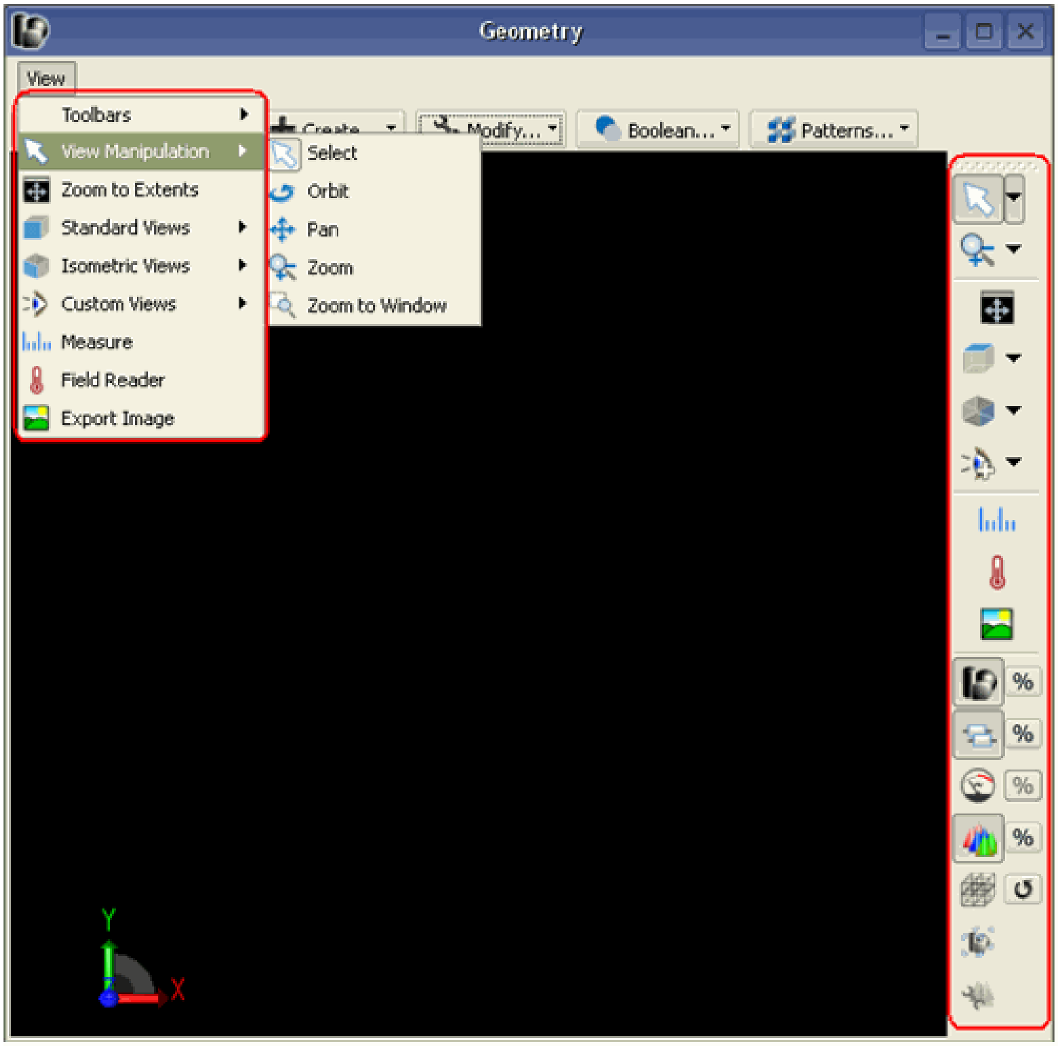
Later sections will describe the operations available in EMPro within *Geometry Tools*. A more detailed discussion of the remaining tools occurs in other sections.

> **Note**
> For more about creating and editing discrete components with *Component Tools*, refer to *Defining Circuit Components and Excitations* (using).
>
> For more about controlling the characteristics of the grid and meshing parameters with *Grid Tools*, refer to *Defining the Grid and Creating a Mesh* (using).
>
> For more about collecting data in EMPro with *Sensor Tools*, refer to *Saving Output Data with Sensors* (using).

View Tools

The *View Tools* are used to alter the perspective of the viewing window by manual rotation, translation, and zoom, as well as automatic orientations to achieve the desired perspective. The View Tools are visible on the right-hand side of the *Geometry* workspace window. They can also be found in the top left-hand corner under the *View* drop-down menu, as seen below.

The View Tools



The *View Tools* may be hidden by right-clicking the toolbar and deselecting the toolbar check-box. The toolbar can be unhidden at anytime using the *View* drop-down menu.

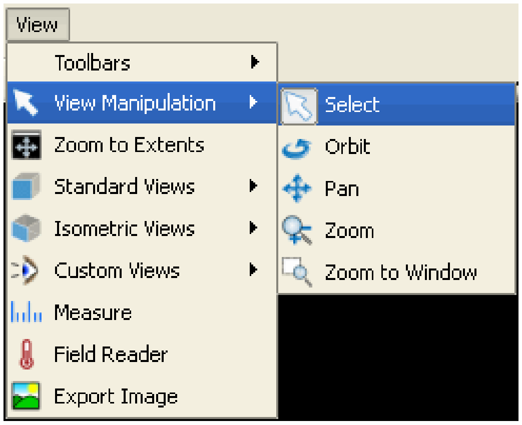
Each tool is detailed below.

View Manipulation

The *View Manipulation* tool provides the following options:

- Select

- Orbit
- Pan
- Zoom
- Zoom to Window



Select

The *Select* tool is the default tool in the *Geometry* workspace window. It is used to select objects as well as manipulate the view of the simulation space.

- Rotation about a fixed point:
  - Left-click and drag.
  - Click the mouse wheel and drag.
- Translation (panning):
  - Right-click and drag.
  - Hold **Shift**, left- or right-click and drag.
- Zooming:
  - Roll the mouse wheel backwards or forwards (to zoom-in or zoom-out, respectively).
  - Hold **Ctrl**, left-click and drag the mouse up or down (to zoom-in or zoom-out, respectively).

Orbit

The *Orbit* tool is selected to perform rotation of the simulation space through left-clicking-and-dragging.

Pan

The *Pan Tool* tool is selected to perform translation of the simulation space through left-clicking-and-dragging.

Zoom

Zoom-in or zoom-out of simulation space by left-clicking-and-dragging the mouse up or down, respectively.

Zoom to Window

Zoom into a rectangular shaped area of the geometry as specified by the user. To use, select the tool, then left-click and drag the mouse to designate the rectangular zoom area.

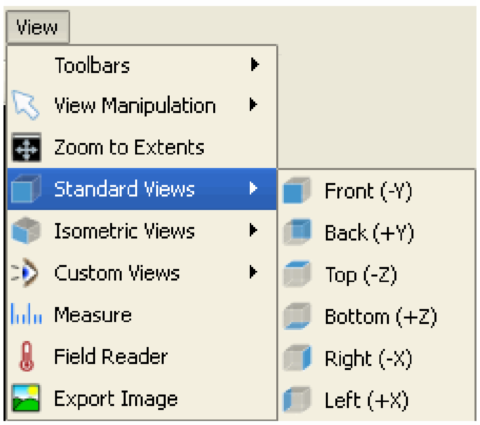
Zoom to Extents

Select this tool to automatically zoom so that the entire geometry can be viewed in the simulation space.

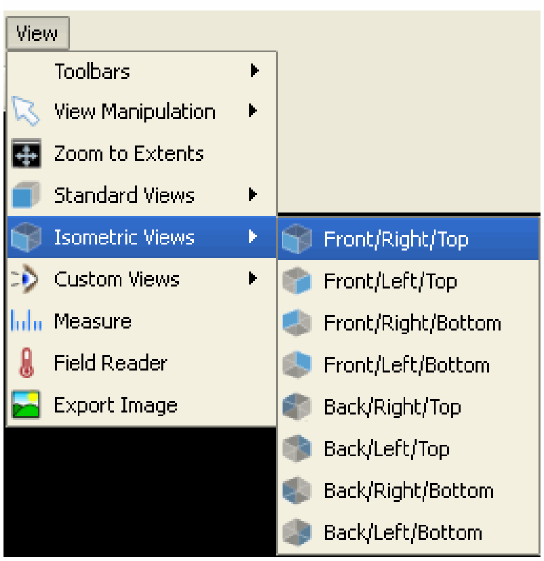
Standard View, Isometric View, and Custom View

The *Standard Views* and *Isometric Views* buttons function to automatically change the perspective of the objects in the *Geometry* workspace window.

Standard Views



Isometric Views



The Standard View changes the view to the following orientations:

- Front (-Y)
- Back (+Y)
- Top (-Z)
- Bottom (+Z)
- Right (-X)
- Left (+X)

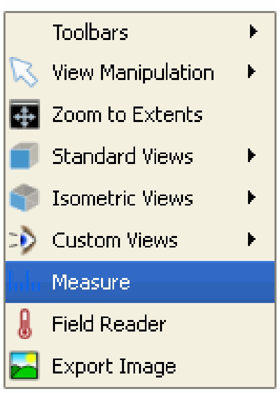
The Isometric View changes the perspective to any combination of these views:

- Front/Right/Top
- Front/Left/Top
- Front/Right/Bottom
- Front/Left/Bottom
- Back/Right/Top
- Back/Left/Top
- Back/Right/Bottom
- Back/Left/Bottom

If these buttons do not achieve the desired perspective, use the *Select*, *Orbit* or *Pan* tools to customize the orientation, and save the desired view by clicking the **Custom Views > Add View** button.

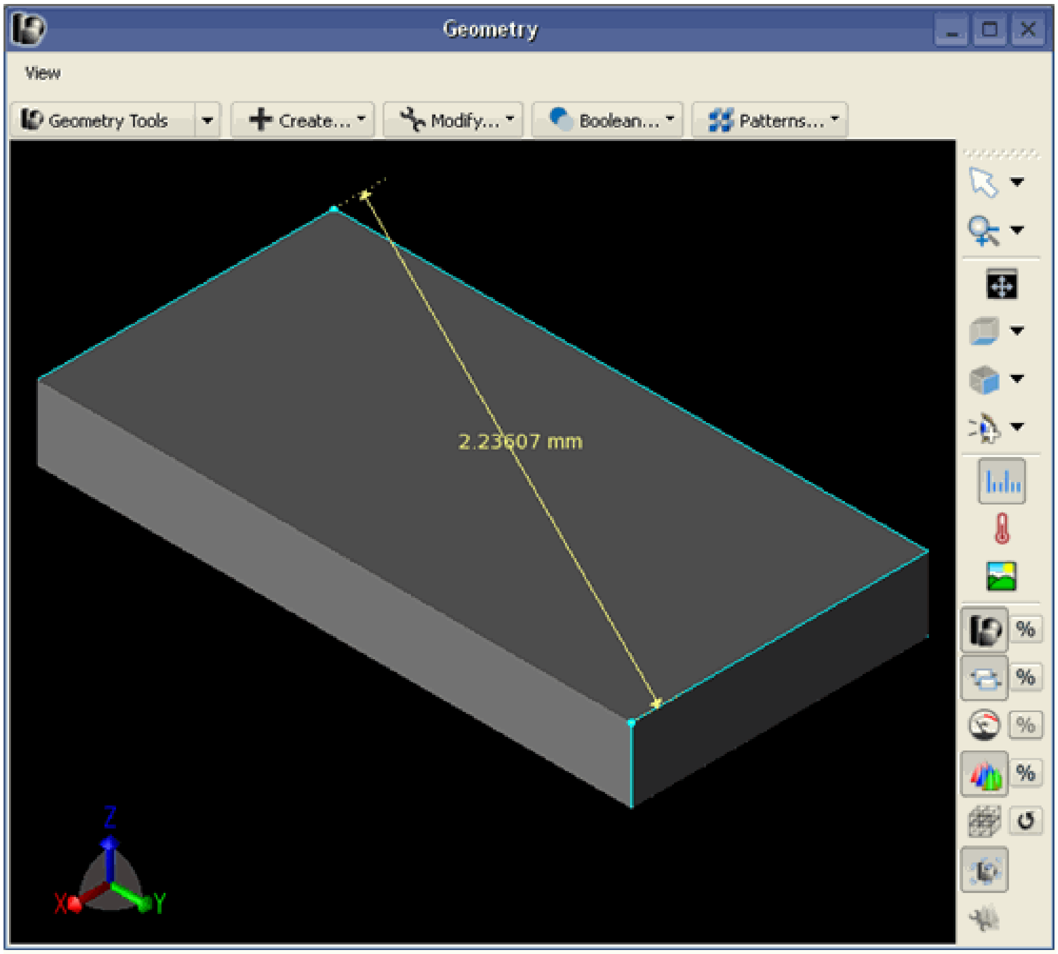
Measure Tool

This tool measures the 3-D distance between any two points by left-clicking a starting point and dragging to an ending point.



The following illustration shows the Measure Tool calculating the distance between the corners of a rectangle.

Using the Measure Tool



Field Reader Tool

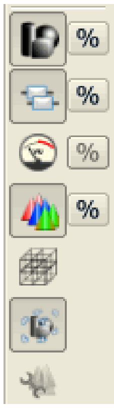
The *Field Reader* tool measures field values at the location where the mouse hovers over the geometry. For more information on the field reader tool, refer to *Viewing Output* (using).

Export Image Tool

The *Export Image* tool takes a screen shot of the geometry as it is currently shown in the *Geometry* workspace window, and saves it to a specified location.

Opacity and Visibility Tools

The *Visibility* buttons control the view of parts of the project.



Clicking any of these buttons will hide its corresponding objects. They include:

- **Parts View** - Toggles the geometric parts on and off.
- **Circuit Components View** - Toggles the circuit components on and off.
- **Sensors View** - Toggles the sensors on and off.
- **Result Fields View** - Toggles the result fields on and off.

Clicking the *Opacity* button located to the right of any button, will bring up a slider to customize the translucency of its objects. The sliders change the alpha of the objects, making them more or less translucent as the slider is dragged right or left, respectively. When the project is in *Mesh View* mode, these buttons are convenient for turning off the view of the solid geometry so that the view of the cell edges is not obstructed.

Note
There are several ways EMPro can render this translucency. For more information on how to adjust these settings the notes on *Transparency Algorithm*, refer to *Application Preferences* (using).

Mesh View

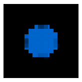
This button toggles between *Mesh View* and Normal View. Alternatively, double-clicking the *FDTD: Mesh* branch of the *Project Tree* will enable *Mesh View*.

When in Mesh View, there are two main viewing modes, *Mesh Cutplanes* and *3D Mesh*, that are controlled by radial buttons along the bottom of the *Geometry* workspace window. A valid mesh must be generated to use these viewing options. For more information on generating a mesh with the *Meshing Properties Editor*, refer to the *Meshing Properties Editor* (using) in "Defining the Grid and Creating a Mesh".

The first mode, *Mesh Cutplanes*, creates cutplanes of the mesh in any or all of the three primary planes. Toggle any of these cutplanes on or off by checking or unchecking their respective boxes. The sliders associated with each of these planes are enabled when its respective plane is turned on. The slider moves the cutplane throughout the slices in the mesh. Additionally, each checked plane will activate the following icons, which aid in manipulating the cutplanes view:

 - Displays all of the geometry/mesh above the cutplane.

 - Displays all of the geometry/mesh below the cutplane.

 - Toggles the view of the grid that occurs normal to the cutplane on and off.

 - Toggles the view of the gridded cutplane on and off.

 - Toggles the view of the electric components on and off.

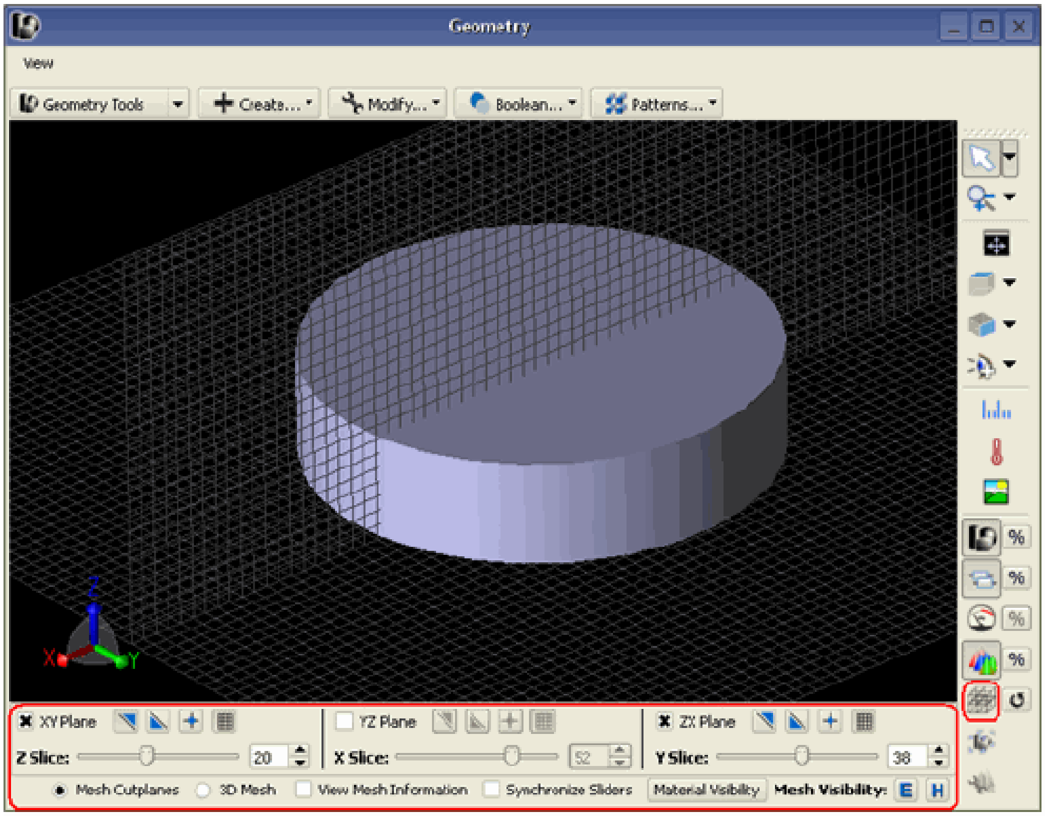
 - Toggles the view of the magnetic components on and off.

There are also two check boxes available:

- **View Mesh Information** - Displays a dialog box with information about the mesh at the location of the mouse.
- **Synchronize Sliders** - Moves the cutplane simultaneously with a slider adjustment while the mouse button is still pressed.

The following illustration shows this first mode that is displayed when the *Mesh View* icon is selected. Note that this is only a preview of the mesh when it is shown while editing the grid within the *Grid Tools* dialog. Any other time, it is a representation of the most recently generated mesh.

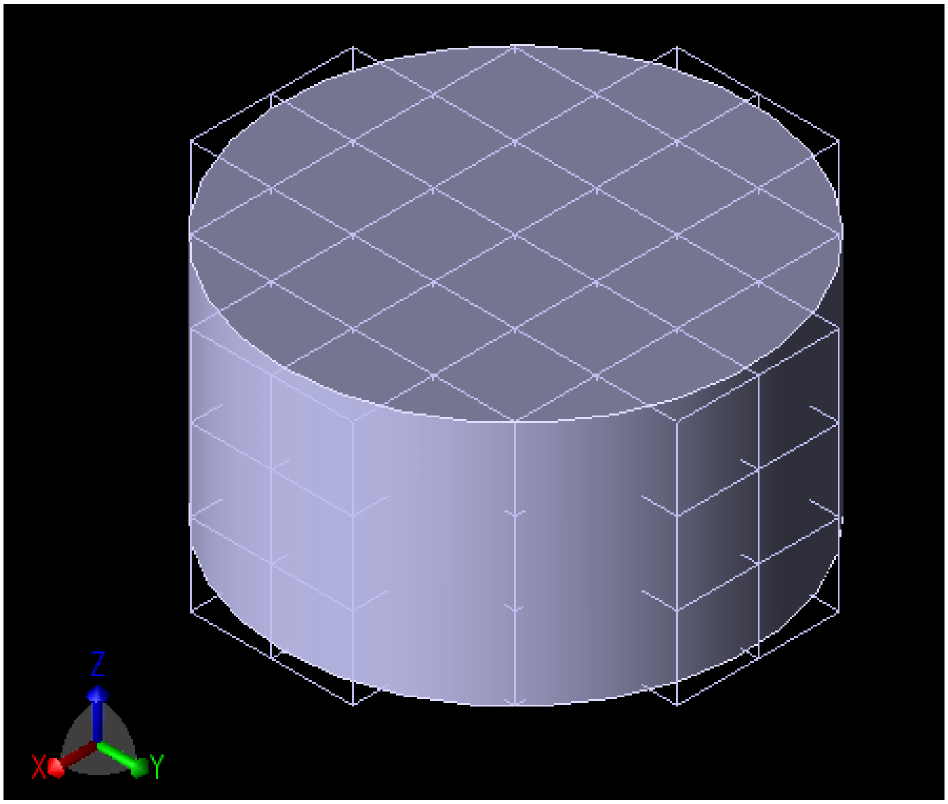
Viewing the mesh



The second mode, *3D Mesh*, provides several different options to view the mesh. Select any of the radial buttons within this option to create a customized three-dimensional view of the mesh. The *Mesh Visibility* icons are available in this mode as well so that *Electrical* and *Magnetic* components of the mesh may be toggled on and off. Additionally, the *Material Visibility* button controls which specific materials are visible in the view.

The figure below shows an object in *3D Mesh* mode with *All Edges* displayed.

Viewing a 3-D mesh



Meshing Options

The *Automatic Remeshing* feature is located in the *View Tools* toolbar. When this feature is enabled, remeshing is performed any time a change is made to the geometry. If this feature is not enabled, remeshing must be performed manually. Automatic Remeshing is not desirable when large meshes are imported because of their memory and performance demands.

The Automatic Remeshing dialog

Toggle Bounding Box Visibility

This button toggles the visibility of the bounding box for the geometry when the geometry is selected.

Toggle Output Viewing Controls

This button toggles the visibility of the output viewing controls for sensor results.

Geometry Tools

EMPro provides *Feature Based Modeling* that allows the creation of geometric objects as a set of repeatable actions rather than one stringent primitive object. This provides more flexibility in customizing an object and allows any unwanted step to be easily undone by use of the *Undo* button without using excess memory that was formally required to rebuild an entire object. It also tracks every step in the modeling sequence as a separate object in the tree to facilitate even simpler additions, deletions and modifications to the modeling sequence.

This section describes the *Geometry Tools* interface, through which geometric modeling in EMPro is performed. This interface enables the user to create new geometry, modify existing geometry, perform boolean operations such as unions, subtraction, and intersections, and create patterns. To begin using Geometry Tools, open the *Geometry* workspace window and select Geometry Tools from the drop-down menu. A more comprehensive discussion of each Geometry Tool is available in the "*Appendix of Geometric Modeling* (using)".

Using Feature-Based Modeling

Parts are created in a step-by-step sequence (such as extrude, revolve, and boolean) that propagate through during creation as the part becomes more complicated. Each step can be re-entered and edited separately.
It facilitates undo/redo operations, parameterization, and constraints

Integrating 2D and 3D Editing

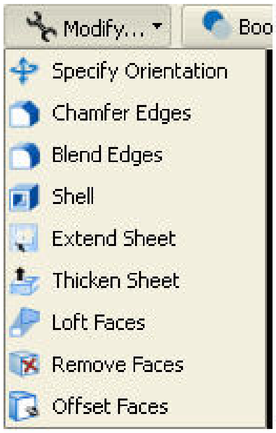
3D objects are created by operations on 2D cross-sections. The global (X,Y, Z) and local (U,V,W) coordinate systems allow easy rotation and translation of objects (connector example).

For more information, refer to *Orienting Objects in the Simulation Space* (quickstart).

Modifying Existing Geometry

You can use the **Modify** menu in the Geometry browser window to modify the geometry of existing objects in the project. This menu provides the following options:

- Create a 3D object
- Translate the object
- Rotate the object
- Chamfer Blend any corner
- Shell Offset Faces from a 3D object



Performing Specific Rotation

Using the Modify menu, you can move and rotate the object with reference to a point:

1. Create an object
2. Select specific rotation menu
3. Move and rotate object

Chamfer edges

For chamfering the edge, perform the followings steps:

1. Create an object.
2. Select Chamfer edges.
3. Perform operation by selecting one edge.

Shell Faces

For shelling the object, perform the followings steps:

1. Create an object.
2. Select Shell Faces.
3. Perform operation by selecting one face.

Performing Boolean Operations

The following Boolean operations need to be tested:

- **Two Parts**: The Two Parts tool provides several boolean operations to subtract, intersect, or unite two objects. To test these feature create two objects, one object must be selected to be the BLANK, and the other the TOOL which acts on the blank.

- **Extrude**: Using the Extrude tool, you can perform an operation on an existing geometry part. In this case, the user chooses the Blank, and then creates the object to use as the TOOL. The user then specifies the orientation of the extrusion and the nature of the operation (Subtract, Intersect, or Union). In essence, this operation is a shortcut for the Two Parts Boolean operation.

- **Revolve**: Using the Revolve tool, you can perform an operation on an existing geometry part. The user chooses the Blank, and then creates the object to use as the Tool. The user then specifies the orientation of the revolution and the nature of the operation (Subtract, Intersect, or Union).

Holes may also be extruded, revolved, or swept through any part any with its respective tool in this menu.

Creating Patterns

Patterns are created by replicating a single selected object multiple times in one of the organized arrangements listed below:

- Linear pattern
- Cylindrical pattern
- Hex-cylindrical pattern
- Spherical pattern
- Elliptical pattern hex pattern
- Radial pattern
- Polar grid pattern

Orienting Objects in the Simulation Space

Within the EMPro interface, there are three primary coordinate systems: Global, Reference and Local. They are distinguished in the GUI as the following reference forms:

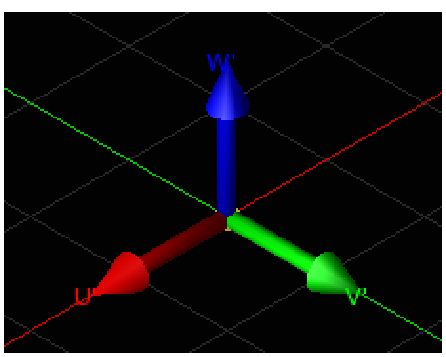
- Global coordinates: *X*, *Y* and *Z*
- Reference coordinates: *U*, *V* and *W*
- Local coordinates: *U'*, *V'* and *W'*

The *Global Coordinate System* does not change throughout the course of a project, so its origin location and three primary axes always remain the same. The 3-D orientation object marked with *X*, *Y* and *Z* in the lower left corner of the *Geometry* workspace window represents the global coordinate system. This set of orientation vectors is referred to as the *Global Triad*.

The *Reference Coordinate System* refers to the orientation and location of objects within their native assembly. If an assembly has not been translated, rotated or shifted, the Reference Coordinate System will be the same as the Global Coordinate System; otherwise, it will be different. Additionally, if an assembly exists within another assembly, its Reference Coordinate System will only consider the location and orientation of its own assembly.

The *Local Coordinate System* is initiated when an editing session begins within the *Specify Orientation* tab. It is initially the same as the Reference Coordinate System, but it changes as translations, rotations, and other adjustments are made to the orientation of the working coordinate system. The figure below shows the set of orientation vectors displayed in this tab, known as the *Orientation Triad*. This coordinate system is reset each time the editing session is reopened, unlike the *Reference Coordinate System*, which is preserved across editing sessions.

The Orientation Triad governs the location and orientation of the Local Coordinate System



The *Orientation Triad* is used to correctly orient objects within the simulation space. The center of the Orientation Triad is referred to as the *Origin*. The Origin can be defined manually by typing in its coordinates, or by clicking the intended origin in the simulation space with the *Pick: Origin* tool. It may also be manually shifted by clicking and dragging the directional axis vector of interest. For more information about the *Pick: Origin* tool, refer to Alignment Tools.

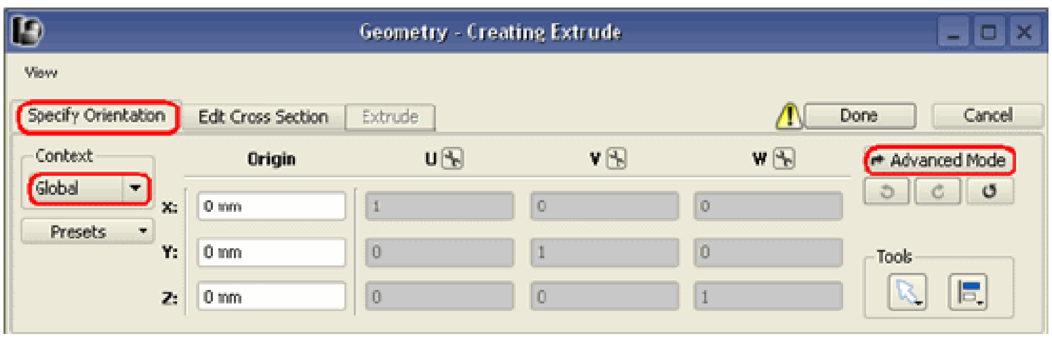
Using the Specify Orientation Tab

The *Specify Orientation* tab in the Geometry workspace is available for positioning any physical object in the simulation space. This tab provides tools for translating, twisting and rotating an object using three different coordinate systems. There are two primary modes that are available within this tab: *Basic Mode*, the default mode which is sufficient under most circumstances, and *Advanced Mode*, which has more powerful functionality. The *Pick* and *Align* drop-down menus available in each mode provide additional options for orienting objects.

Basic Mode

The next figure shows the **Specify Orientation** tab in *Basic Mode*. The *Context* controls whether the Orientation Triad's position and orientation is cited in the Global or Reference Coordinate System. Notice that in this figure, the coordinates are all defined in terms of *X*, *Y*, and *Z*, denoting the Global Coordinate System.

The Specify Orientation tab in Basic Mode

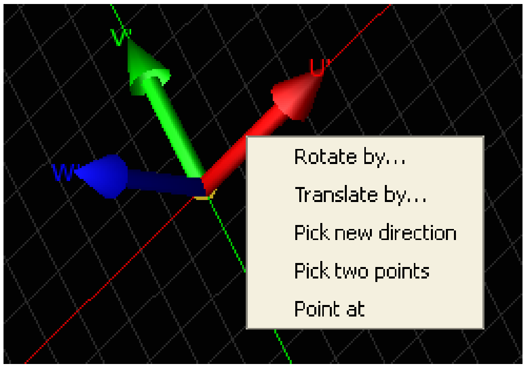


The origin and orientation of the *Orientation Triad* can be adjusted in several ways within Basic Mode. All of these methods are also available in *Advanced Mode*.

The *Origin* coordinates specify the location of the origin of the Orientation Triad. The origin can be updated manually, or can be adjusted by clicking and dragging on one of the colored primary-axis vectors of the Triad. The location display will update automatically.

The orientation of the axes is defined by clicking the icon found next to any of the axis definitions. This will change the column from gray (read-only) to white, so that the values in the column can be edited. The orientation may also be adjusted by right-clicking the axis and choosing the appropriate tool in the context menu, displayed below.

Customizing the orientation of the simulation space



- **Rotate By** - rotates the coordinate system about the chosen axis.
- **Translate By** - prompts the user to type in a translation distance along the selected axis (analogous to clicking and dragging on the axis vector).
- **Pick New Direction** - redefines the direction of the selected axis in a new direction specified by the user. This tool is also useful to align the selected axis with the surface of other objects in the simulation space.

Note
Clicking a point in the simulation space will assign the direction vector directly into the space. Thus, adjusting the view with the *View* buttons may facilitate this assignment.

- **Pick Two Points** - aligns the axis to the direction of the vector between two user-selected points.
- **Point At** - redirects the selected axis by directing the vector from its origin to a user-selected point.

Clicking the *Adavanced Mode* in the Basic Mode dialog window will bring up the *Advanced Mode* window.

Alignment Tools

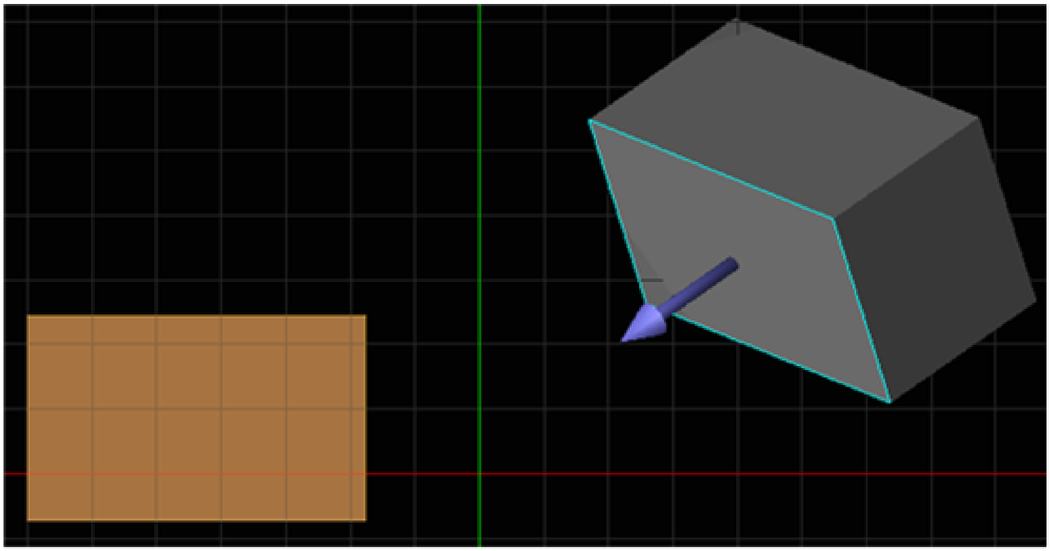
Direction Picking Tools

The *Direction Picking Tools* menu provides six tools for defining the location and direction of the *Orientation Triad*. In the descriptions of tools used to align one object to another, *reference object* refers to the pre-existing, stationary object used as a reference for the alignment, and "object to align" refers to the object that is to be aligned to the reference object using the respective tool.

Simple Plane

Select a plane on a reference object to orient the plane of the object to align. The normal vector will be directed out of (orthogonal to) the selected plane, as shown in the figure below. EMPro will adjust the object to align so that its face is aligned with the selected plane of the reference object. Press the |**Space**| key until the object is correctly oriented.

Aligning a new object with an existing reference plane



Origin

Select a point anywhere in the simulation space to position the origin of the Orientation Triad. (The normal vectors will not change orientation.)

Normal

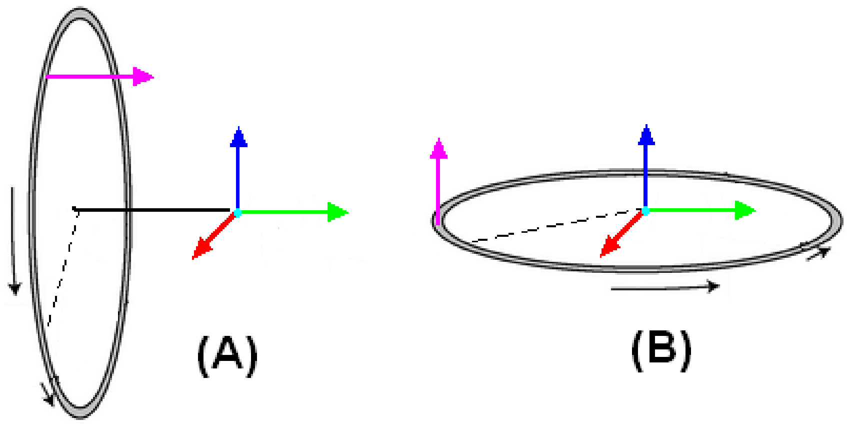
This tool enables you to adjust the direction of the normal vector of the Orientation Triad while maintaining the same origin point. The normal direction can be defined by clicking a point in the simulation space, in which case the *W'* - normal vector will be positioned directly into the simulation space. The normal vector can also be aligned with the face of a reference object by holding the mouse over one of its faces, and the normal vector of the object to align will be aligned with this face of interest.

- It is convenient to use the *View* buttons to position the simulation space so that this placement achieves the desired orientation.
- Additionally, you can select **|Space|** to reverse the direction of the orientation vector before placing it.

Direction for Twist

Twisting about an axis is performed by defining a normal vector. The object will be twisted about the origin as the twist slider is moved left and right. The figure below shows the placement of the normal (purple arrow) and the radius of twist about the origin.

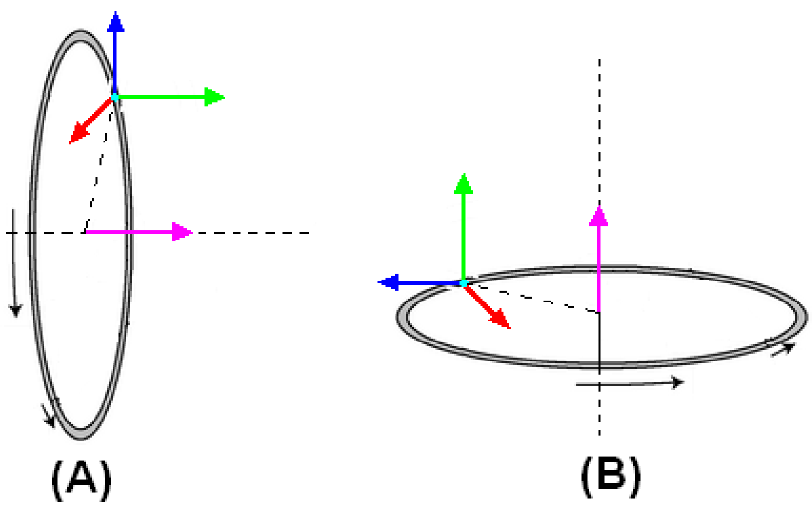
Twisting an object about the origin



Axis for Rotation

Unlike the *Direction For Twist*, the *Axis For Rotation* moves the object and *Orientation Triad* about a user-defined axis. The object will not be twisted as it is rotated (i.e., a face originally faced in the *Z* -direction will always face in that direction independent of its displacement).The figure below shows the rotation of the Orientation Triad and object about the user-defined axis (shown as a purple arrow).

Rotating an object about a user-defined axis



Axis for Rotation and Twist

This tool is analogous to the *Axis For Rotation* tool except when the object is rotated around the picked axis, it will also be twisted.

Align menu

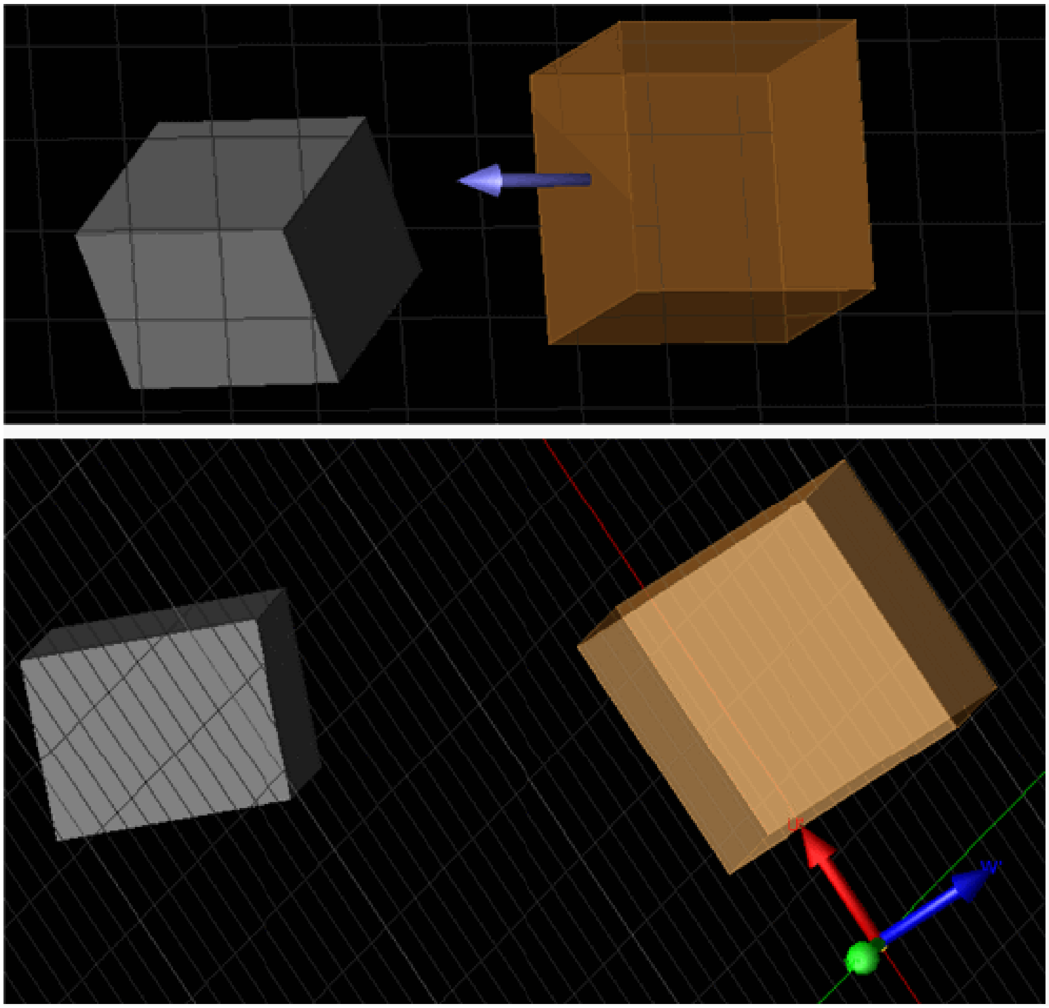
The *Align* menu provides several tools for aligning features of a new object with those on a reference object.

Align Face Normals

This tool aligns the face normal of the object to align with the normal of the reference object. Pick the face on the object to align, then pick the face of the reference object. The two normal vectors will be used to orient the coordinate system such that the normal vectors are pointing toward each other or away from each other. The following figure shows an object to align (left) being aligned with a face on the reference object (right). The bottom image shows the two objects after they have been aligned.

- Select |**Space**| to reverse the direction of the orientation vector before placing it.

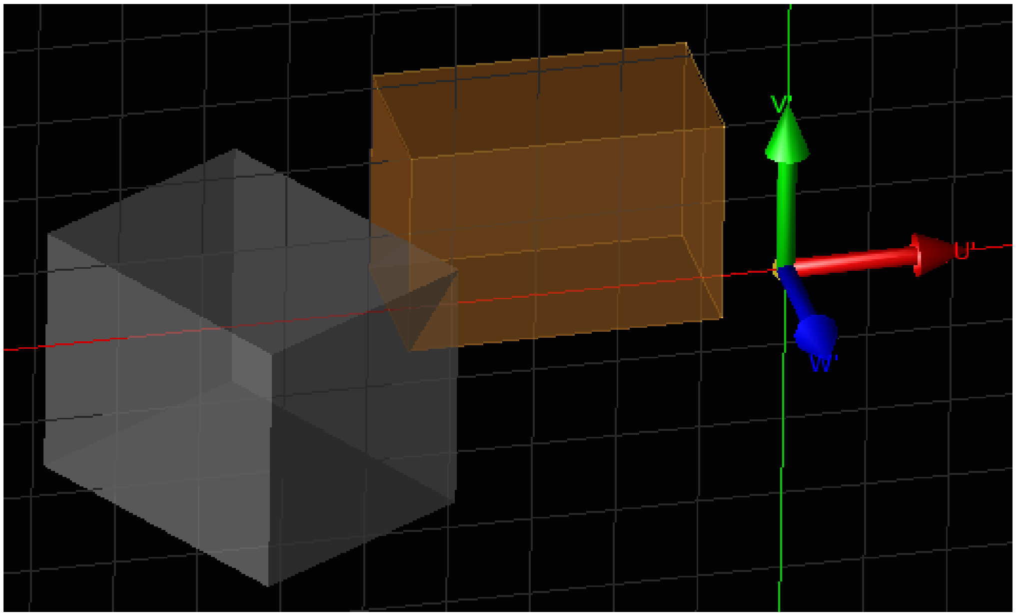
Aligning two face normals



Match Points

This tool matches a user-selected point on the object to align with a user-selected point on the reference object. Select a point on the object to align and select a second point on the reference object. The following figure shows the original objects in the next figure after two of their corners were matched.

Aligning points on objects



Match Faces

This tool functions much like the *Match Points* tool above except that it aligns the normals of two objects rather than discrete points. Select the object to align, then select the face of the reference object. The two positions will be used to translate the first part such that the two selected points are coincident. Since aligning the faces of two objects do not give regard to the placement of the moved object, it is often useful to slide the moved object along an axis on the Orientation Triad until the desired placement is achieved.

- Press |**Space**| to reverse the direction of the orientation vector before placing it.

Advanced Mode

The options available in *Advanced Mode* enable you to manipulate virtually all aspects of the orientation of the coordinate system. The three main areas for editing in this mode, *Anchor*, *Axis*, and *Twist*, are located on the left of the editing window. Within each of these three areas are also definitions for *Translations* and *Rotations*, which are additional offsets to the geometry within the defined coordinate system.

Translations can also be applied manually in any direction within the *Translation* definition. A translation will not affect the location of the anchor point, only the location of the Orientation Triad and attached object. It is important to be aware of which of the three coordinate systems is selected in the *Context* definition when applying a manual translation, since it may affect the proper placement of the geometry.

Similarly, rotations may also be manually defined within the *Rotation* definition. The rotation, across the angle specified within *Angle*, will be applied to the Orientation Triad (and attached object) at the anchor point, in the direction specified by the *U'*, *V'*, and *W'* definitions. For example, defining:
U' = 0
V' = 0
W' = 1
Angle = "PI rad"

The *Anchor* will be rotated Π radians about the *W'* axis. If the definitions were changed to:

U' = 1
V' = 0
W' = 1
Angle = "PI rad"

then *U'* and *W'* will be automatically adjusted to *U'* = "0.707107" and *W'* = "0.707107", since the resulting rotation will rotate the orientation 0.707107 radians in the *U'* and *W'* directions.

The net movement of the Orientation Triad in the coordinate system will be:

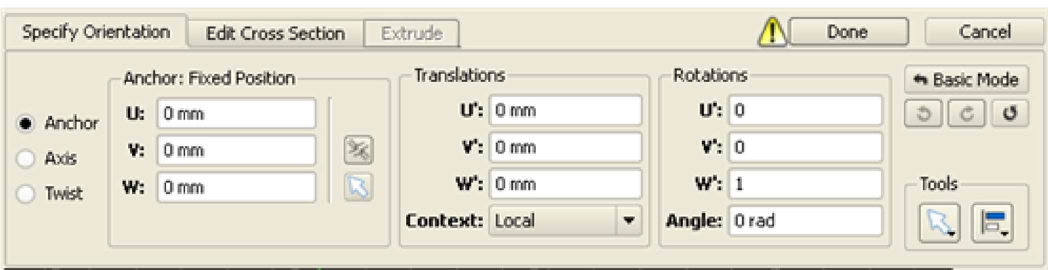
$U + U' + U'_{rotation}$ $V + V' + V'_{rotation}$ $W + W' + W'_{rotation}$

The *Anchor*, *Axis*, and *Twist* buttons are detailed below.

Orienting the Anchor

The following figure shows the *Anchor* editing dialog within *Advanced Mode*.

Modifying the location and orientation of the anchor

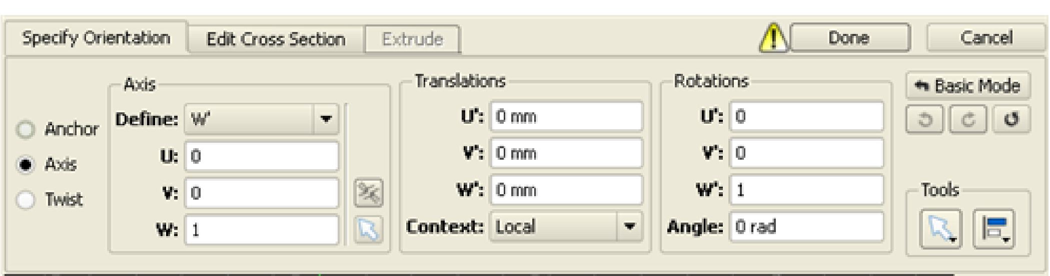


The anchor point is represented by a blue dot in the simulation space. It remains in place regardless of rotations and translations that are applied to the Orientation Triad. When this dialog is first opened, U, V, and W within the *Anchor: Fixed Position* section represent the location of the anchor point in the Reference Coordinate System. Adjusting the location of the anchor point will adjust the location of the *Orientation Triad* as well (although moving the Orientation Triad will NOT affect the position of the anchor point).

Orienting the Axis

The next figure shows the *Axis* editing dialog within *Advanced Mode*.

Modifying the orientation of the axis



The *Axis* dialog controls the orientation of the axes of the anchor. The default orientation of the *Orientation Triad* vectors are:

U': *U* = 0
V': *V* = 0
W': *W* = 1

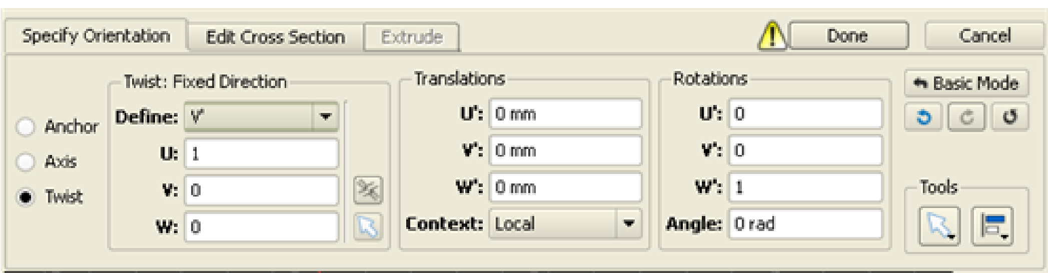
These can be redefined to assume any orientation. Changing the orientation of one axis will automatically adjust the orientation of the other two axes, since they must always exist at 90 degrees from one another.

The *Define* drop-down list enables you to specify which axis is associated with the direction defined in the *U*, *V*, and *W* orientation boxes.

Twisting the Axis

The twist axis defines a second direction for orthogonalizing the triad. The next figure shows *Twist* editing dialog within Advanced Mode.

Twist dialog



Using the Picking and Detach Tool

The Picking Tool and Detach Tool buttons enable you to use the mouse to select an anchor point, axis location, or twist location, or to detach an anchor, axis location, or twist location (if defined), respectively. If you select the Detach button, FDTD removes the anchor point, axis location, or twist location from the reference geometry it is associated with. This change removes the connection between the objects. For example, when two objects are anchored together they will both move if the orientation of either object changes. If the anchor is removed, a change in orientation to one object will no longer affect the second object.
The Picking Tool is always active. However, the Detach Tool is active only when an anchor point, axis direction, or twist direction is defined using the reference geometry.

The Picking and Detach tools for the Anchor, Axis and Twist areas operate independently based on the radio button selected in the Specify Orientation Tab (Anchor, Axis, Twist). For example, if the Axis radio button is selected, the Picking Tool will enable you to place an axis on an object and the Detach Tool (if active) will enable you to detach the existing axis from an object. You cannot select or detach any existing anchor points or twist directions when the Axis radio button is selected, only Axes.

Anchor is a position, axis and twist are directions. Therefore, depending on which of the radio buttons is active you'll be doing a different thing. When you're picking a position you only see a dot and you can choose a position on a line, vertex or face. When you're picking the axis or twist you're defining a direction relative to other existing geometry. You'll see an arrow to show you which direction will be picked. When picking a position or direction you have extra options available which can be seen by mousing over existing geometry. Depending on what the mouse is over a different tooltip will appear telling you what actions are available. You can cycle through directions using space. Both position and direction have the ability lock to a part or hide a part. Picking a position can center on a face or use the bounding box.

Orienting 2-D Sketches on the Sketching Plane

The correct placement of the Orientation Triad in relation to its corresponding parts is critical to some 3-D modeling operations, especially during revolution and sweeping operations. It is important to understand the functionality of the Orientation Triad in relation to the operation that is performed. The following two examples show how the Orientation Triad should be placed about a 2-D cross section.

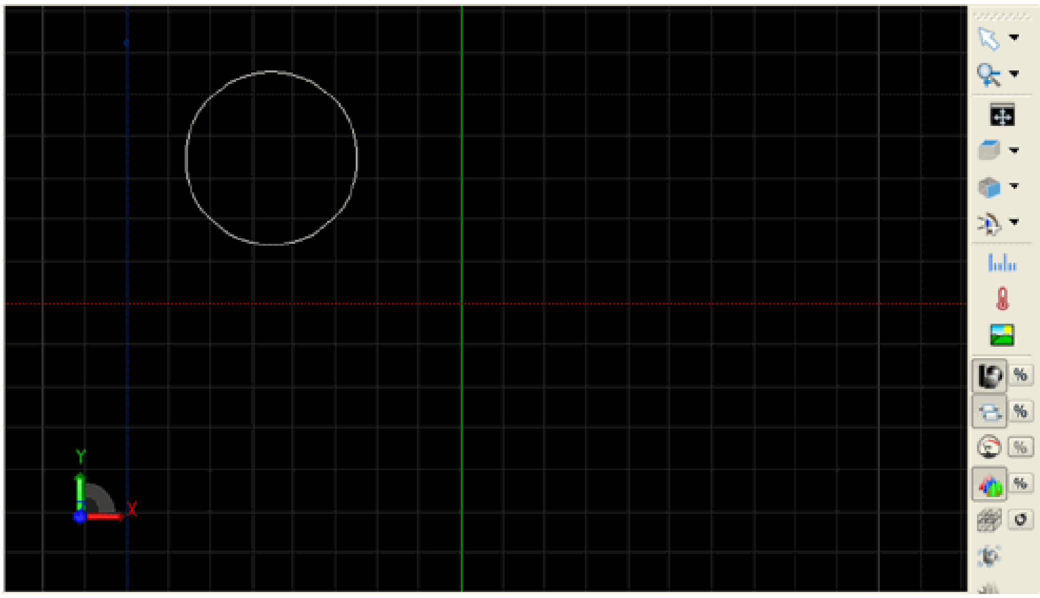
Orienting Sketches for Revolutions

The following illustrations demonstrate the placement of the Orientation Triad during a revolution operation. The first illustration shows a simple 2-D cross section sketched in the *Edit Cross Section* tab. Notice that the sketch does not intersect any of the primary axes.

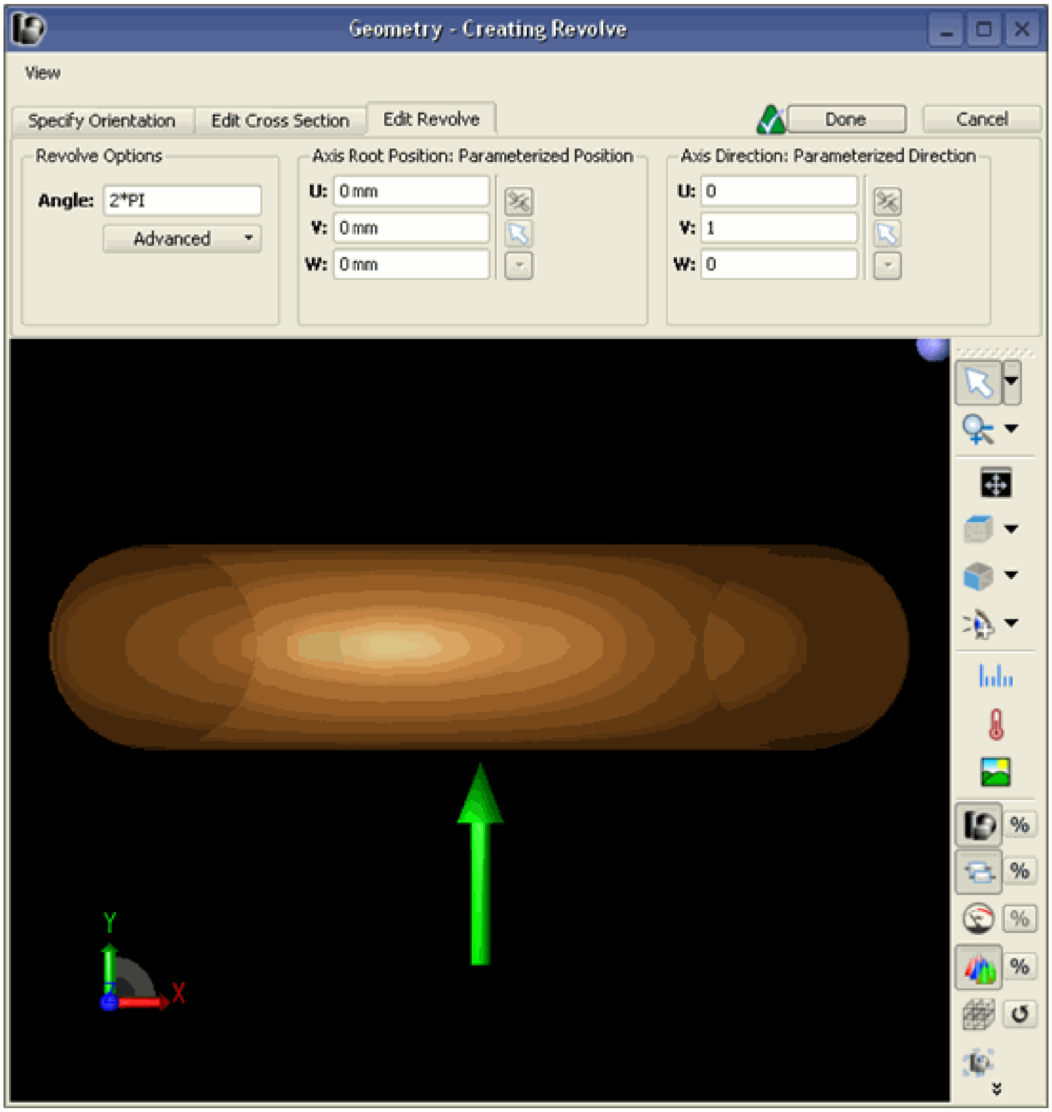
> **Note**
> It is important that the 2-D sketch does not touch any axis of revolution, or else the revolution cannot be performed.

The next figure shows the revolution of the circle about the *V'* -axis, after which we see the revolution of the circle in the *U'* - and *V'* -directions. Finally, in the last illustration we see how the revolution operation is used to create a hollow, or solid sphere from an open or closed semi-circle cross section, respectively.

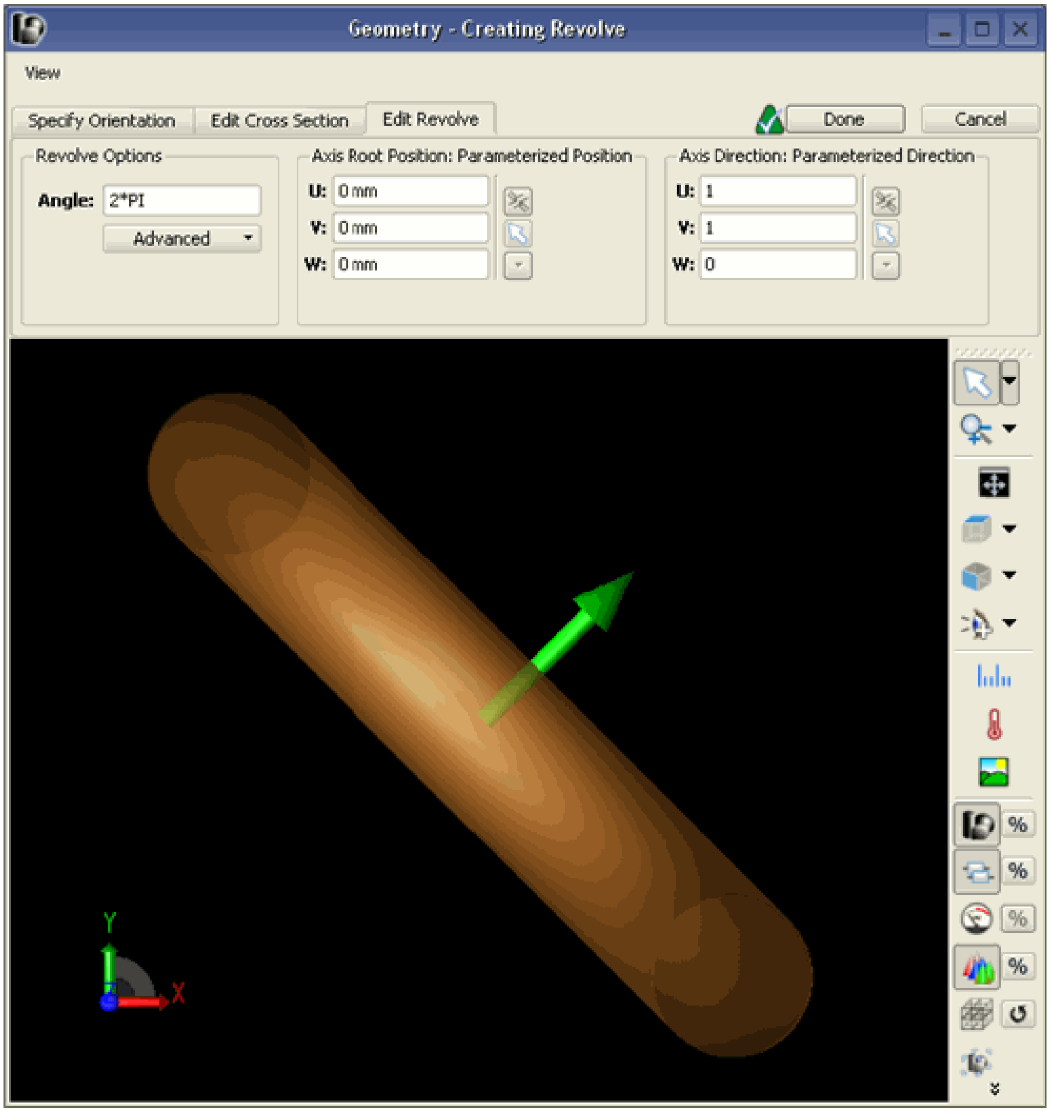
Orienting a 2-D cross section about an axis of revolution



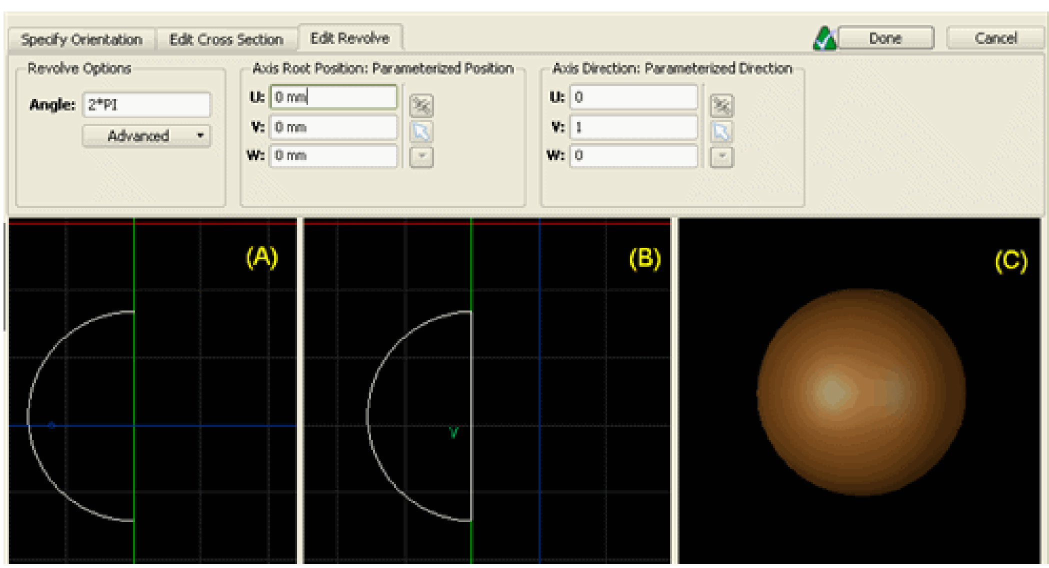
Cross section revolved about the *V'* -axis



Cross section revolved about the *U'* - and *V'* -axes



Using (A.) open (hollow) and (B.) closed (solid) cross sections to (C.) create a sphere with the Revolution operation



3D Library Components

EMPro consists of a built-in library of parameterized 3D objects that includes the following types of shapes:

- Box
- Cylinder
- Piramid
- Ring
- Sphere
- Cone
- Helix

To insert the shape, click on the symbol at the bottom of the EMPro window. You can also edit the parameters prior to entering the object or modify them after insertion. The object can be positioned after insertion by using the **Specify Orientation** menu.

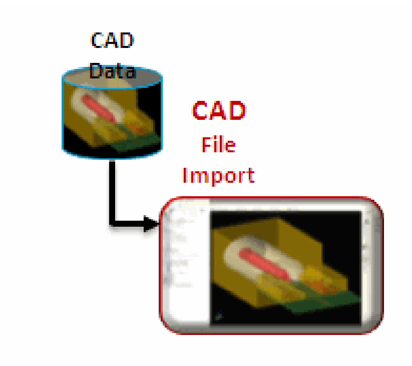
If a new parameter name is used that does was not defined in the **Parameters** menu in EMPro, the new parameter name will automatically be added.

Importing Files

In order to enhance the geometric modeling process, EMPro has the ability to load both CAD drawings, voxel objects and mesh objects.

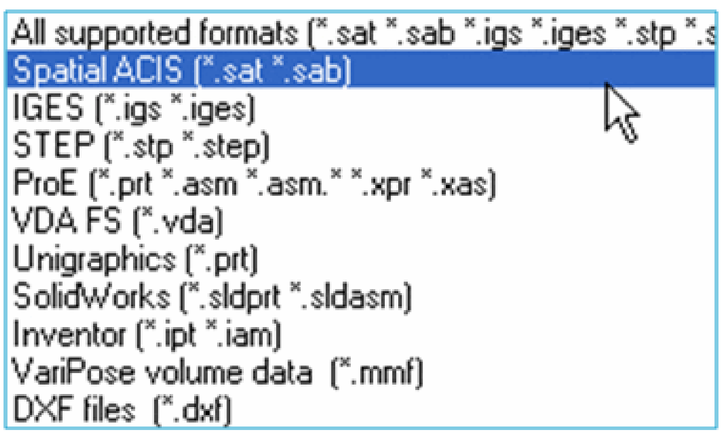
CAD Files

EMPro allows you to import various industry standard CAD formats. This is one of the fast way to bring existing design geometries in the EMPro simulation space.



CAD Files can be imported and exported in various file formats in EMpro, as shown in the following figure:

Figure: Supported CAD Formats

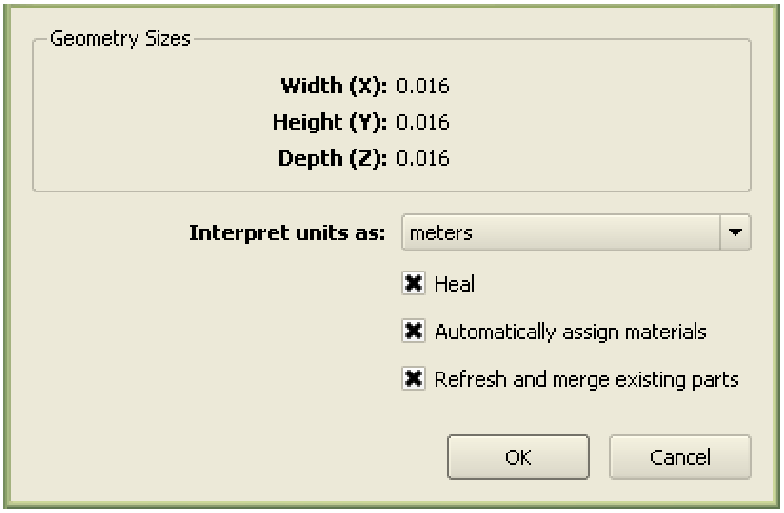


CAD Import Options

The CAD Importer is used to import CAD files from many popular modeling packages into EMPro for use in simulations. To import a CAD file, select **File** > **Import** > **CAD File(s)** and load the desired file.

After the you select the CAD file to load, a dialog box will pop-up with several important options. This is shown in the following figure.

CAD import options

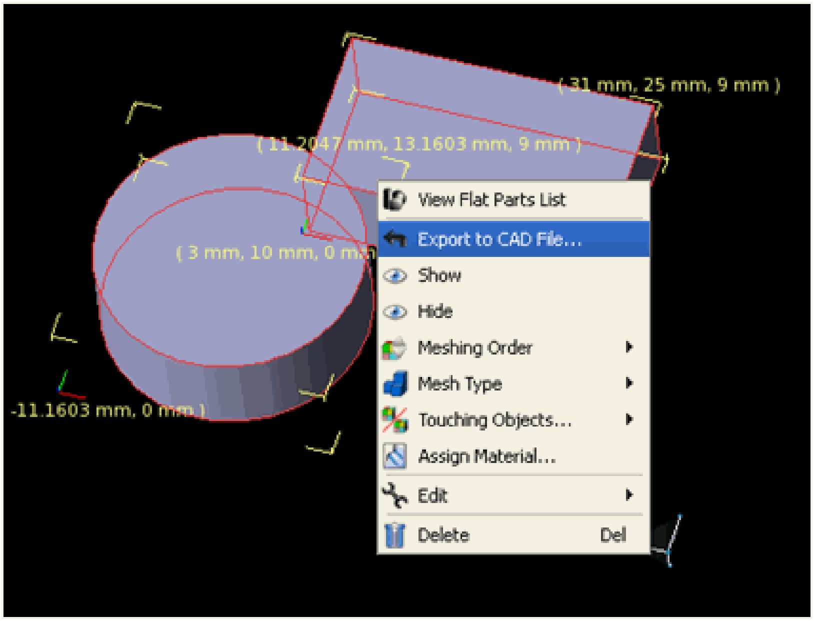


- The *Interpret Units As* drop-down list assigns the units to the CAD file after it is imported.
- The *Heal* check-box will check imported files for errors and correct them as needed. In particular, objects imported from *IGES* files and *STEP* files may have errors. For more complex objects, this can be a time consuming process and pop-up window will display the progress of the operation.
- The *Automatically Assign Materials* check-box appears after the external CAD files have been read. When this option is selected, color information will be extracted from the imported parts if it is available. If a material exists in the project that has the color of the imported part, that material is assigned to the part. If no material is found, and a color is available, a new material is created and assigned.
- The *Refresh And Merge Existing Parts* check-box should be checked when the user has already loaded a CAD file into an EMPro project and desires to update it with a newer (external) version of the source file. When this option is selected,
  - parts that are *used* within your EMPro project will be updated with any geometrical changes present in the newly imported CAD file.
  - parts that are *new* to the imported CAD file are added to the project.
  - parts that have been *deleted* in the imported CAD file, but are still present in the EMPro project, will remain in the project without change.
  - parts that have been *deleted* in the EMPro project, but are still present in the CAD file, will be added to the EMPro project with *Meshing Disabled* and its *Visible* property unchecked.

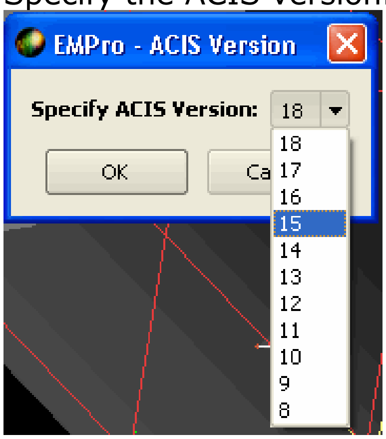
> **Note**
> In each of the above four cases, all changes from the original files will be documented in a shortcut group in the *Groups* branch of the *Project Tree*.

Exporting a CAD File

1. Create a 2D or 3D geometry in EMPro.
2. Right-click the object and select **Export to CAD File**. This opens the *Export to CAD File* dialog box.
3. Specify a file name and save it as a SAT File. You can also export as an IGES or STEP file.

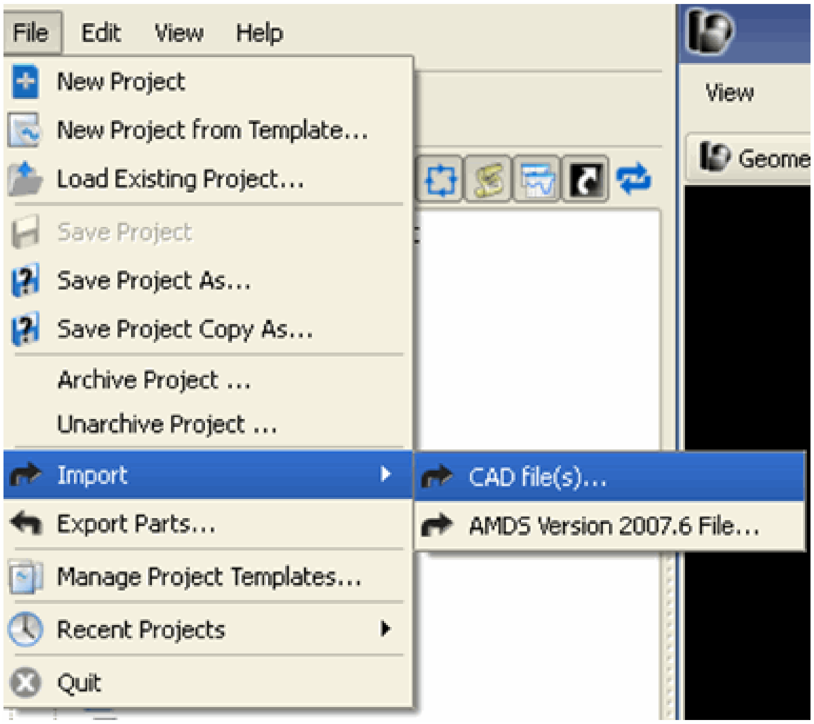


4. Click **Save**.
5. Specify the ACIS version.

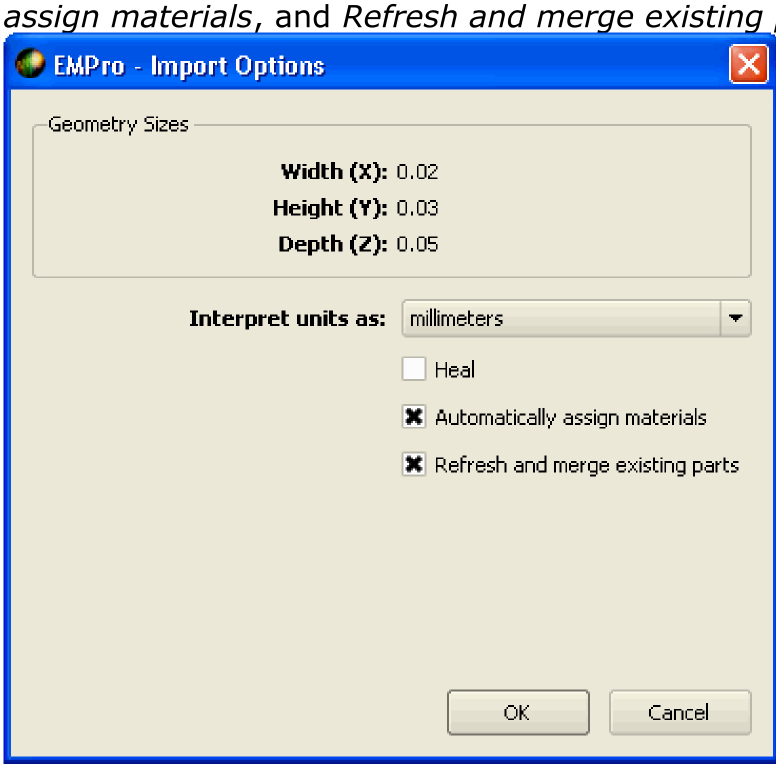


Importing CAD Files

1. Select **File > Import > CAD File(s)**.



2. Point to the directory where the required files are located and select the required files.
3. Click **Open**. This opens the *Import Options* dialog box.
4. Set the **Interpret units** as **millimeters**. Do not select the *Heal*, *Automatically assign materials*, and *Refresh and merge existing parts* options.



5. Click **OK**.

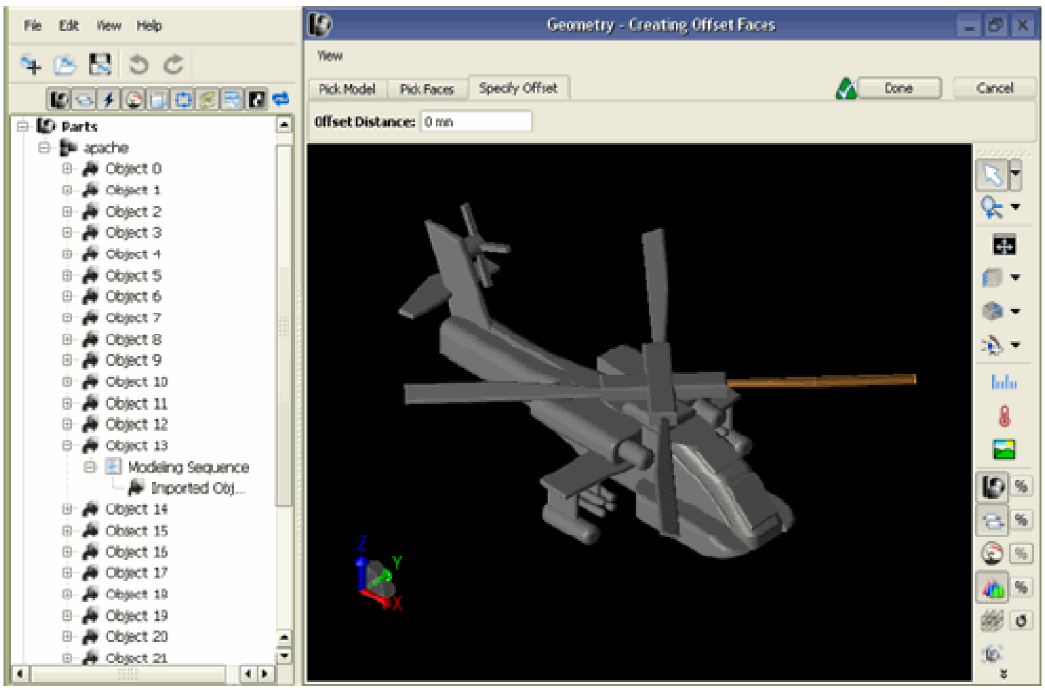
Note
Imported objects are assembled to an assembly part with a name *Multi-file import* and do not have materials assigned to them. You need to rename the imported objects for a better readability of object names and assign them to certain materials.

Modifying a CAD File

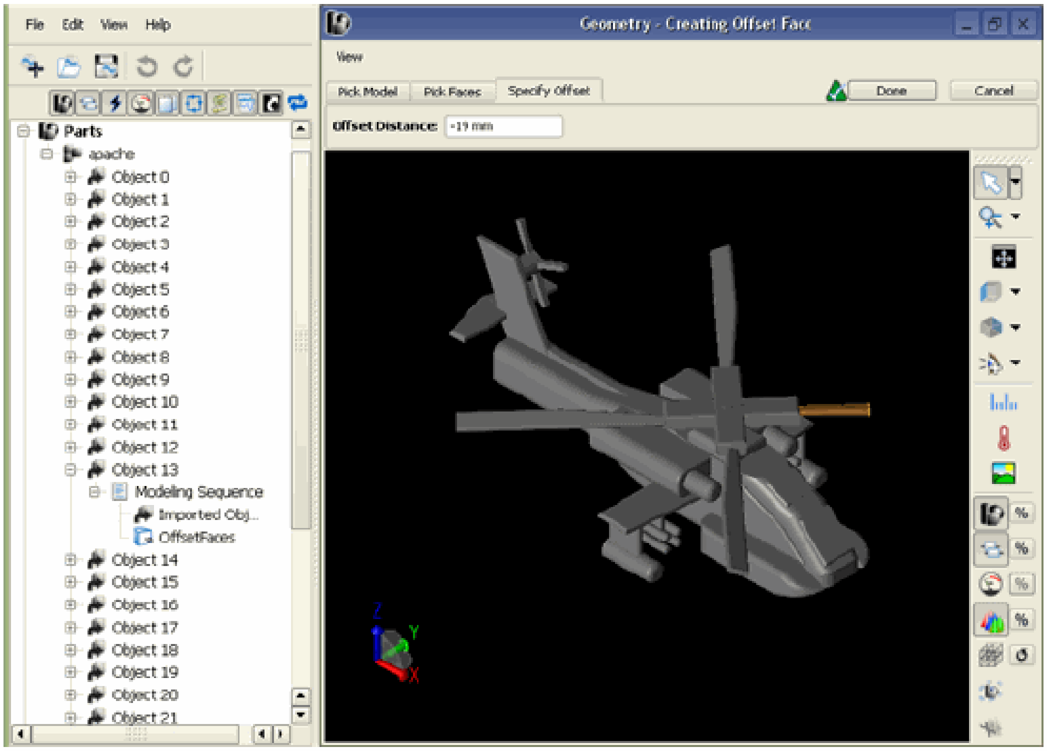
Once imported, an assembly containing all of the parts of the CAD file is added to *Parts* branch of the *Project Tree*. Since every part of the CAD file is treated as its own separate object, all available modeling operations can be applied to any individual object imported from the file. Selecting an operation in the *Modify* drop-down box within *Geometry Tools* will enable you to select any part to modify.

Below, the first figure shows an imported CAD object before a modification operation is applied. the second figure shows the CAD object after an *Offset Faces* operation is applied to one of its parts, and the resulting *Modeling Sequence* object that is added to the tree.

CAD file before modification



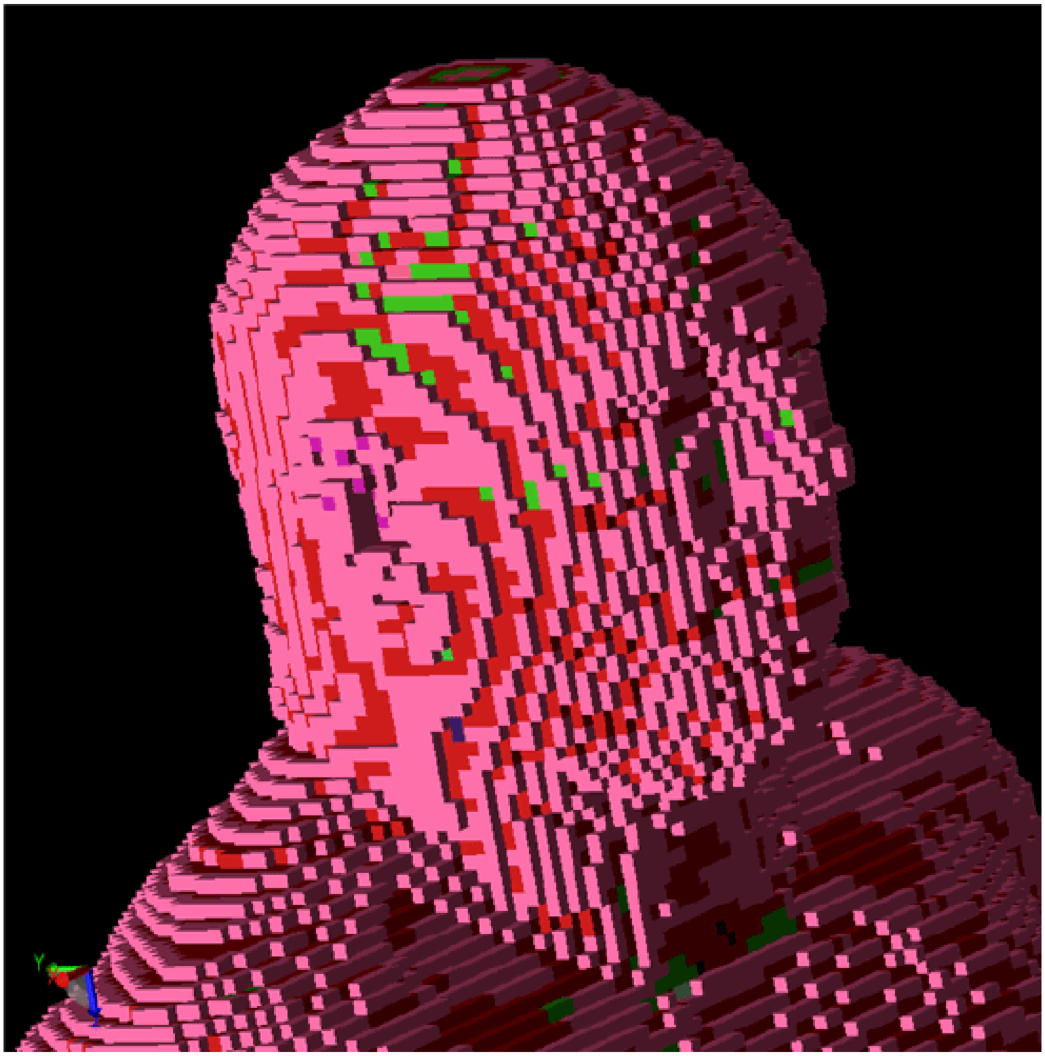
CAD file after modification



Voxels

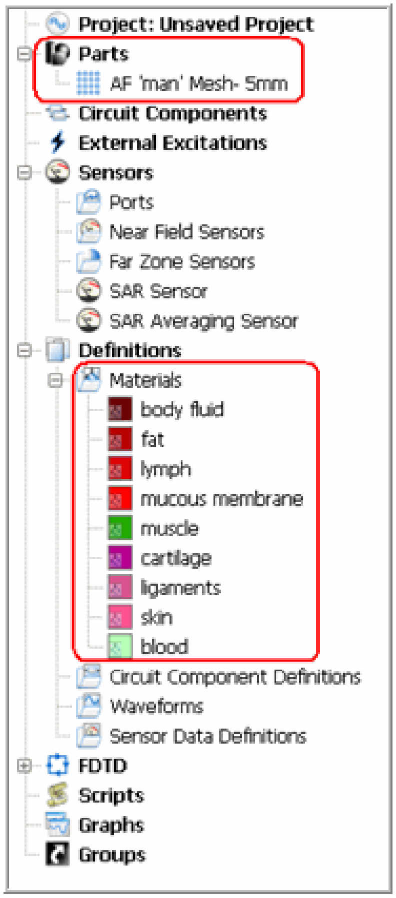
Voxel objects are volumetric pixel points in space with specific volume characteristics. Voxels are a similar to CAD objects in that they are linked to an external voxel data file, and are loaded through selecting the select **File** > **Import** > **CAD File(s)** option. The external voxel data file follows the format specified from the *VariPose .mmf* file. The illustration below displays an imported **.mmf* file.

The head of an imported human body mesh comprised of 5-mm voxel objects



The following illustration shows the *Project Tree* after the the **.mmf* file seen above was imported. Note that an object has been added to the *Parts* branch that contains the voxel object and a list of all the materials contained in the object have been added to the *Definitions: Materials* branch.

The Project Tree with imported voxel object



Meshing a Voxel object

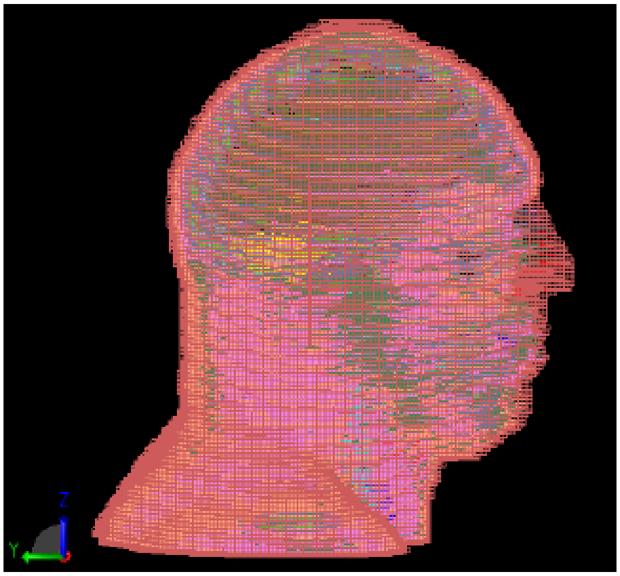
To set the meshing parameters for the voxel object, right-click the object in the *Project Tree* and select *Meshing Properties* to open the *Meshing Parameters Editor*.

> **Note**
> For more on configuring the settings of the *Meshing Parameters Editor*, refer to *Volume Meshing Options* (using) located in "Defining the Grid and Creating a Mesh".

Mesh Objects

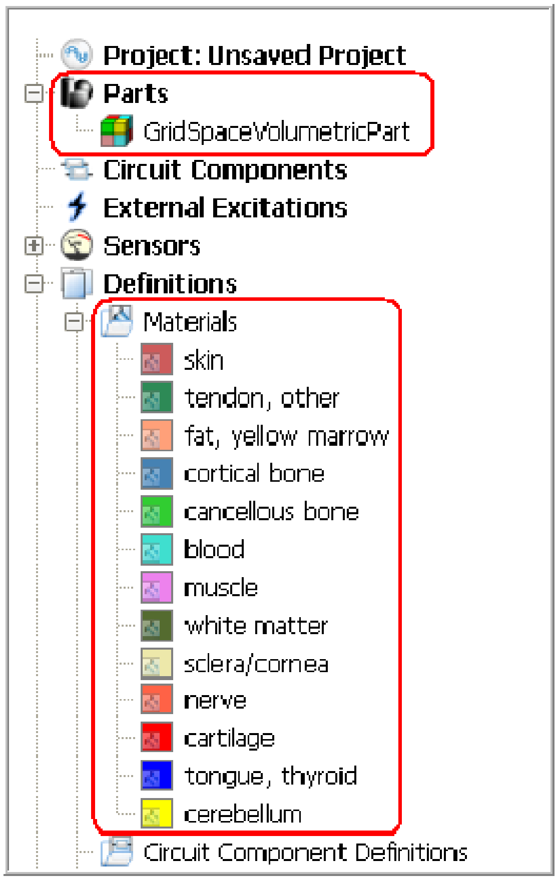
A mesh object is essentially a collection of edges (of various sizes) with applied materials. It is a subsection of the EMPro grid from a previous version. Mesh objects, like voxel objects, are linked to an external mesh object data file, and are loaded through the *File > Import > AMDS Version 2007.6 File...* option. The following figure displays an imported *.mesh file.

The head of an imported human body mesh object



The next figure shows the *Project Tree* after the the **.mesh* file seen above was imported. Note that an object has been added to the Parts branch that contains the mesh object and a list of all the materials contained in the object have been added to the *Definitions: Materials* branch.

The Project Tree with imported mesh object



Like a voxel object, to set the meshing parameters for a mesh object, right-click the object in the *Project Tree* and select *Meshing Properties* to open the *Meshing Parameters Editor*.

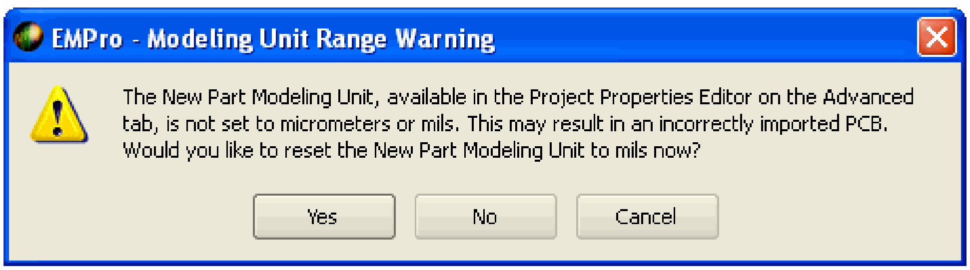
Note
For more information on configuring the settings of the Meshing Parameters Editor, refer to *Volume Meshing Method* (using).

Importing ODB++ Files

The FDTD simulation method can be used to simulate the way energy propagates through and around Printed Circuit Boards (PCBs). To create complex PCB designs, geometry can be imported from another application.
The ODB++ format is used to import complex geometry as it is a standardized format which facilitates data exchange between many applications in the PCB industry.
EMPro reads the ODB++ database and constructs a three dimensional representation of the data.

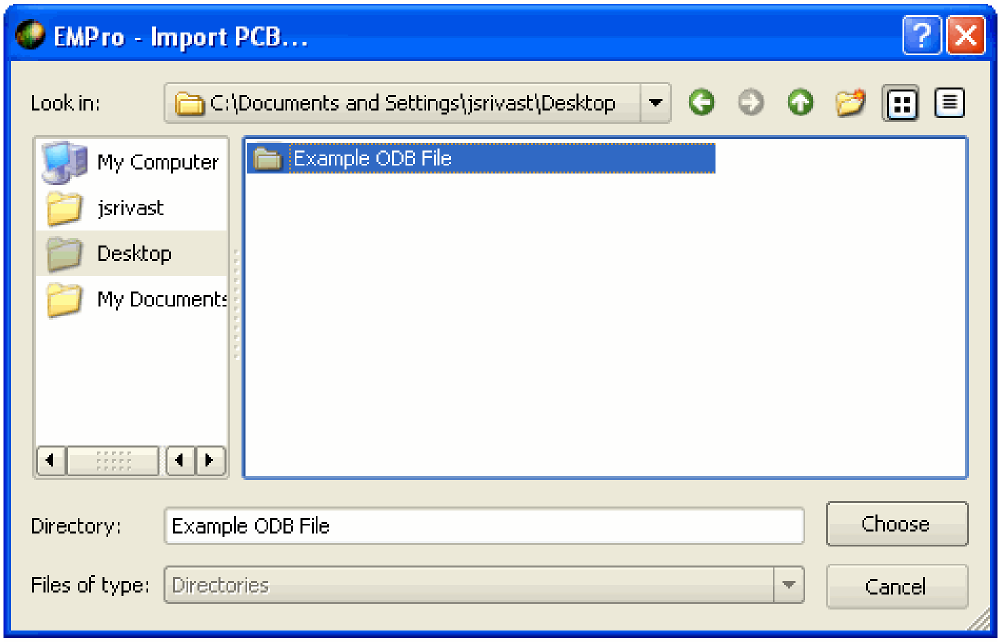
To import ODB++ files:

1. Choose **File** > **Import** > **PCB**. The *Modeling Unit Range Warning* dialog box may appear.

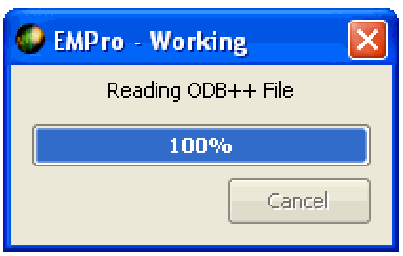


Note
To successfully construct the solid model, the New Part Modeling Units must be in mils or micrometers. If the Project Properties Editor is not set to one of these two units, EMPro will prompt to change the unit.

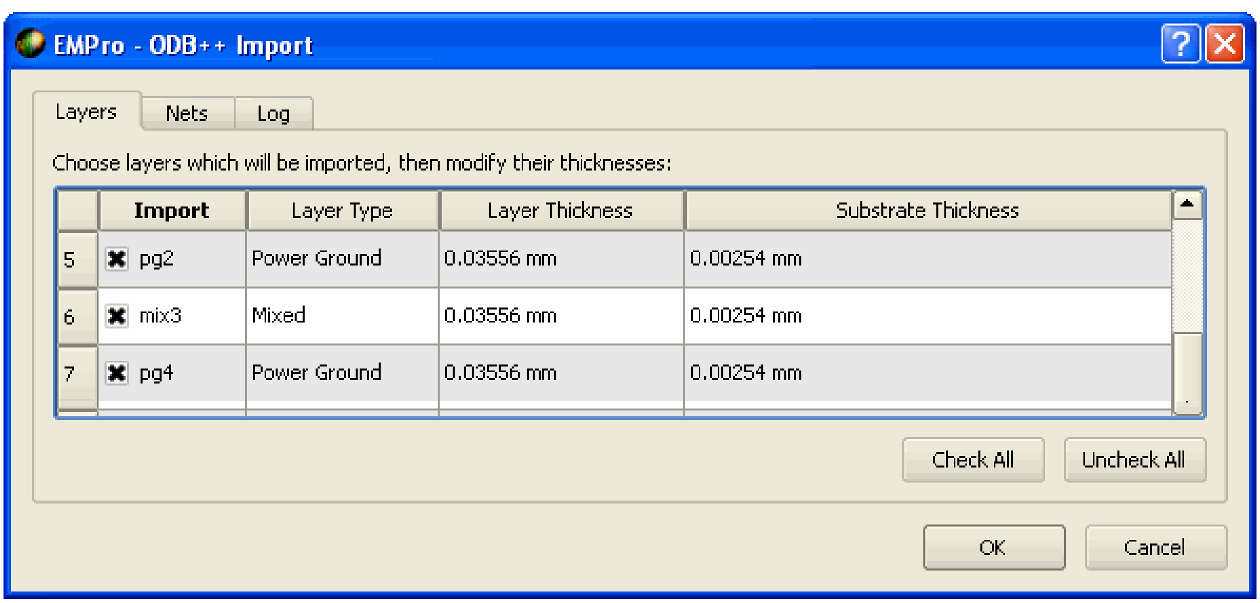
2. Click **Yes** to convert New Part Modeling Units to mils or micrometers else Click **No**. After the unit conversion is complete, the *Import PCB* dialog box appears.



3. Select the ODB++ file to be imported and click **Choose**. A Status window will display the reading status information of the ODB++ file.



Once the ODB++ file is completely read, the *EMPro-ODB++ Import* dialog box is displayed.



In the *EMPro-ODB++ Import* dialog box,

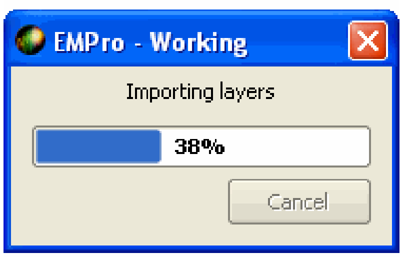
The **Layers** tab displays a list of all the layers in ODB++ file. By default, all the layers are imported. Layers of the types signal, power ground, drill and mixed are checked by default, but each layer can be changed by clicking the check box next to the layer name. To import selective layers, un-check the layers from Import column under Layers tab. The Layer Type is a non-editable field and is displayed only for information purpose. You can modify the Layer Thickness and Substrate Thickness fields. Each thickness can be modified by clicking on the value in the table and entering a different value.

Note
You can check or un-check all the layers using Check All and Uncheck All buttons, respectively.

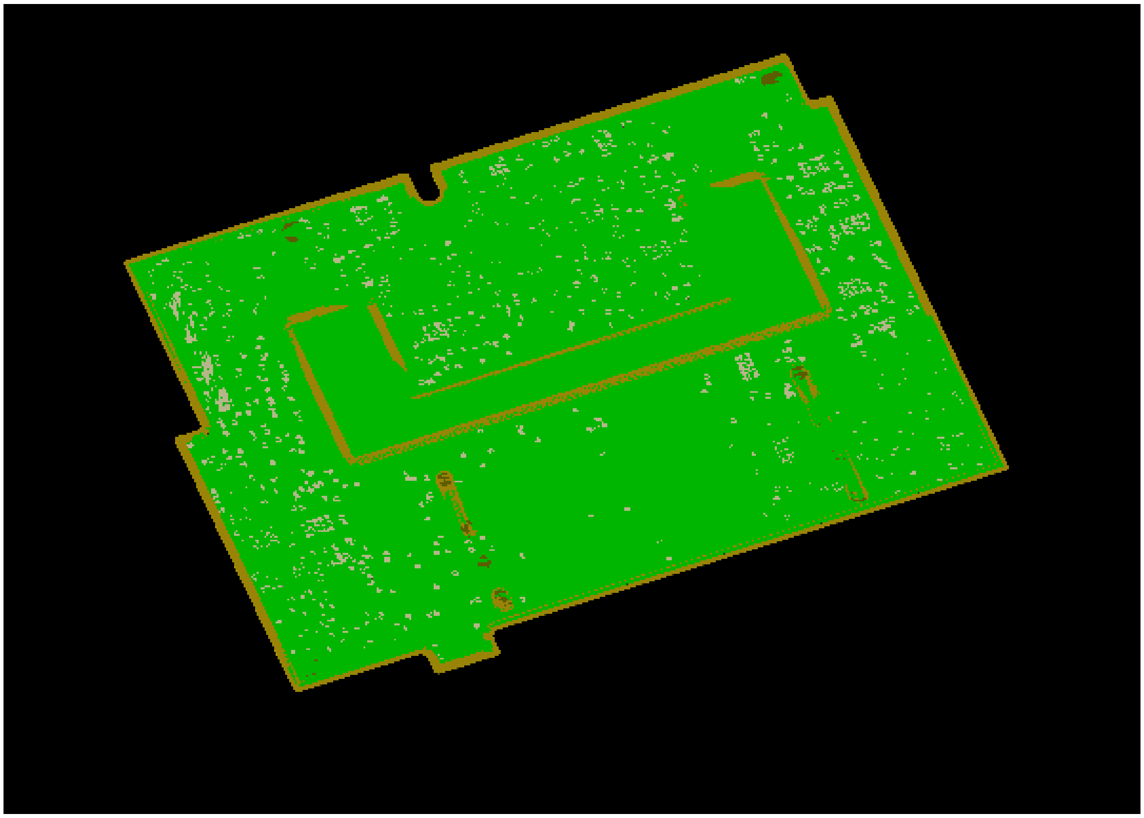
The **Nets** tab is available if the ODB++ database contains EDA data. If there is no EDA data or no nets are chosen, the geometry for each layer is separated into pads, traces, surfaces and polarity. If nets are chosen, the geometry for each net is separated in its own part, and then the remaining geometry is separated into layers.

The **Logs** tab provides the feedback generated when the data is read from the ODB++ database. The contents of this log indicates any errors encountered. The messages provided in the log do not include the status of converting the data to the solid models. After you chooses OK, the data is converted into parts. In the event of an error during this conversion, a window will appear after conversion has finished indicating the problems encountered.

4. Once you are done with layer selection, click **OK** to import ODB++ file into EMPro else click **Cancel**.
   A Status window opens displaying the import information.



5. Once the file import is complete, the output data is displayed in the *Geometry* workspace window.



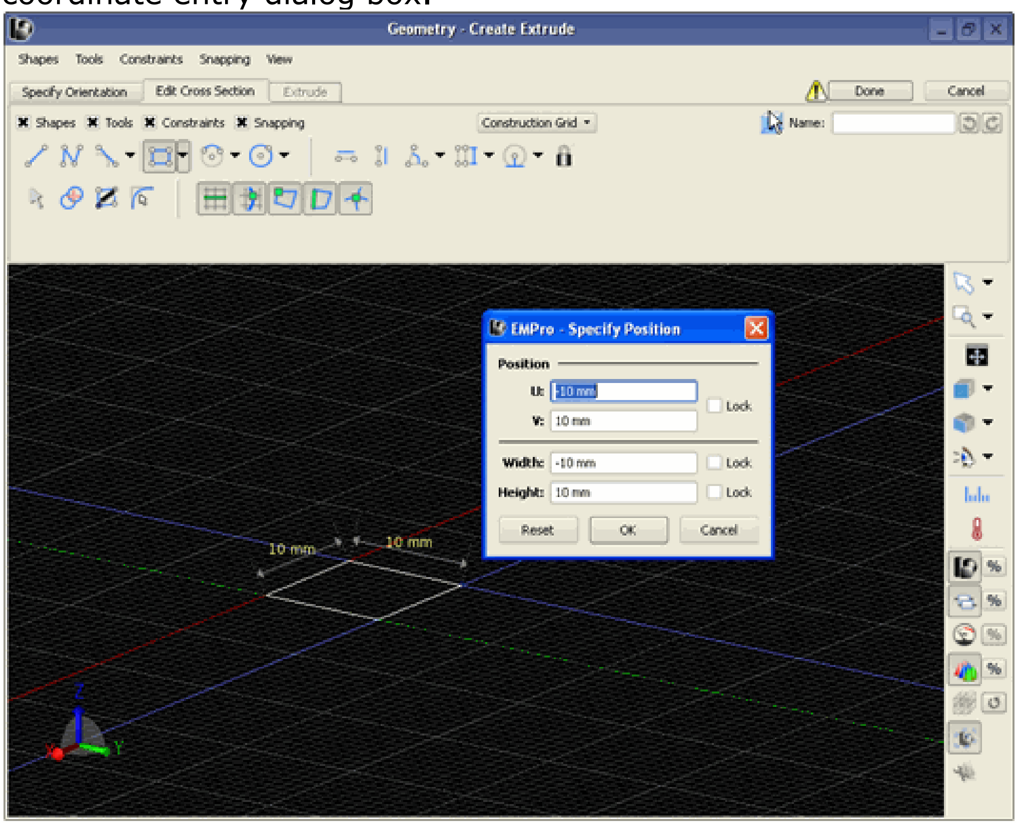
Modifying Existing 2D and 3D Objects

The modeling operations applied to the object are stored in EMPro. This enables you to change the operations according to your requirements. You can modify existing geometries which include imported objects, for example, move, copy, rotate, and Boolean operations. This section describes how to modify 2D and 3D objects.

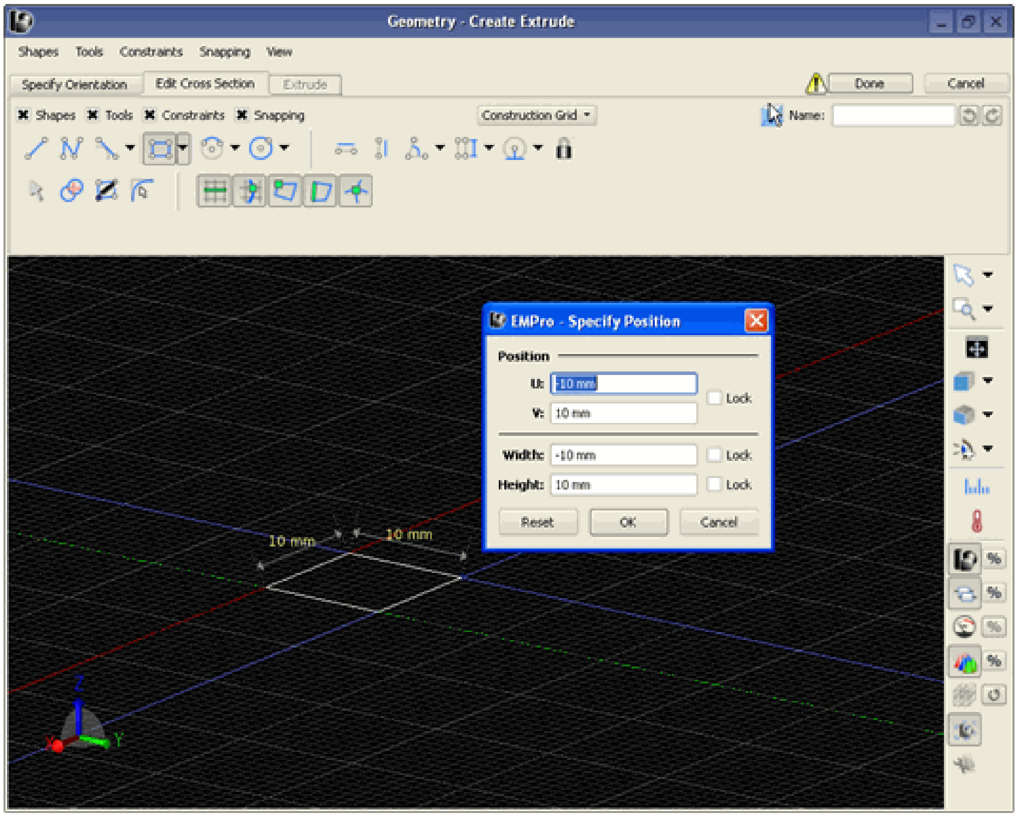
Creating 3D Objects from 2D Objects

Perform the following steps for creating 3D objects from 2D objects:

1. Select **Create** > **Extrude**.
2. Specify the **Name** of the object.
3. Set the orientation of the drawing plane. By default, XY plane is the drawing plane orientation.
4. Draw 2D objects such as circle, rectangle, or polygons.
5. Select (0,0) from the lower-left corner of the object or press **Tab** to activate the coordinate entry dialog box.

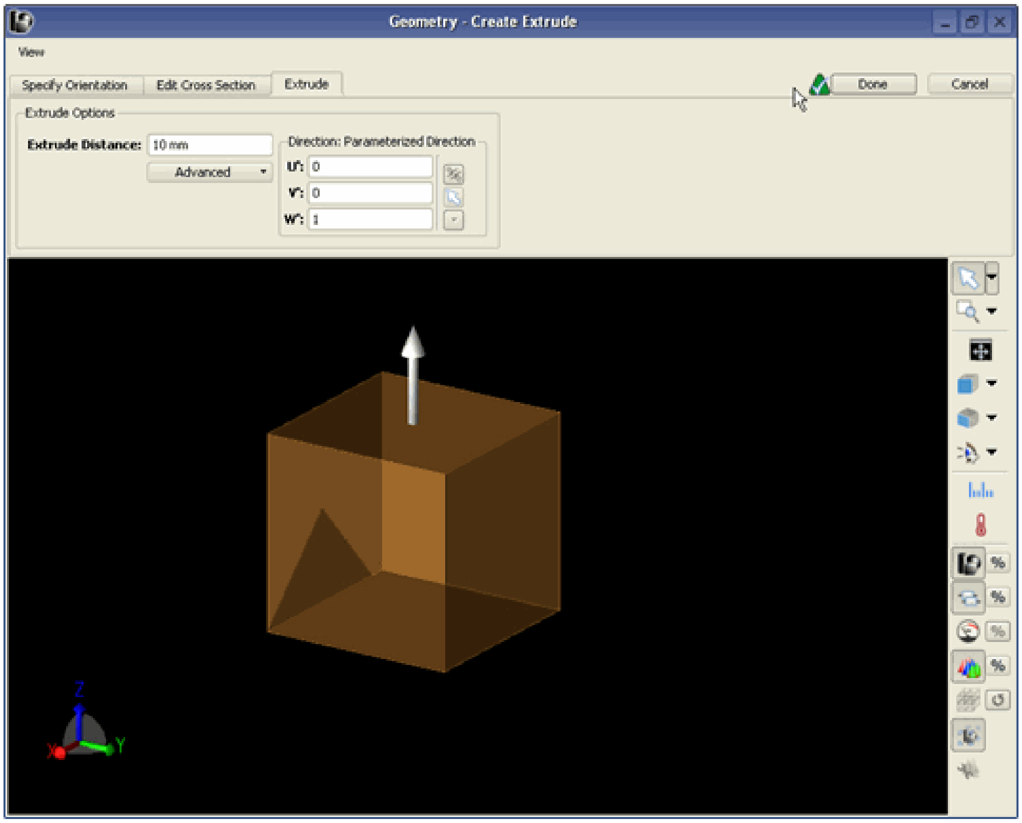


6. Extrude it to create a 3D object. Select the required coordinates or press **Tab** to open the **Specify Position** dialog box.



Note
When you move the the mouse over the drawing plane, the dX and dY (or U and V) values are displayed.

7. Click **Ok**.
8. Select the **Extrude** tab.



9. Enter a value in the in the **Extrude Distance** text box. You can also move the arrow

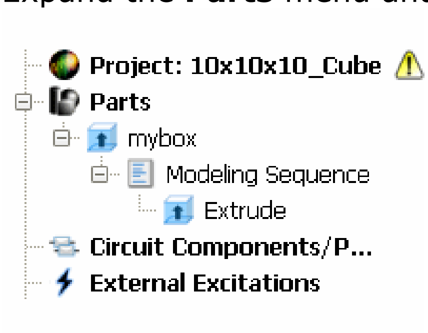
in the geometry space to change the distance.

10. Click **Done**. The green check mark means that there is no problem with this object creation, as shown below:

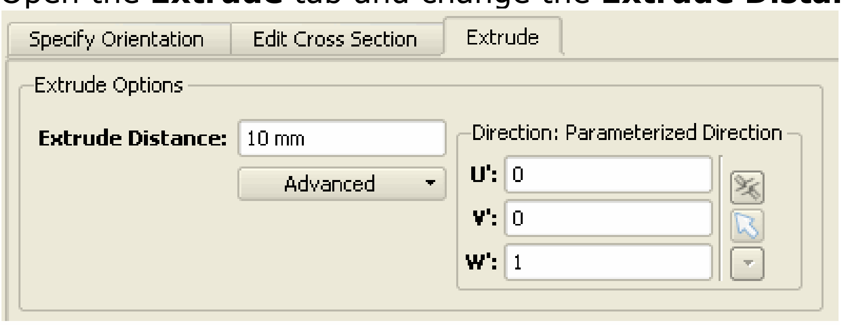
Resizing Existing 3D Objects

Perform the following steps for resizing the height of an object:

1. Open a EMPro project.
2. Expand the **Parts** menu and double-click **Extrude**.



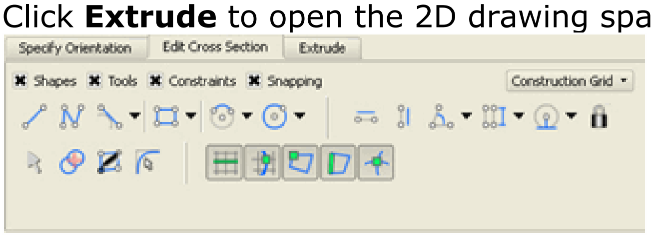
3. Open the **Extrude** tab and change the **Extrude Distance**.



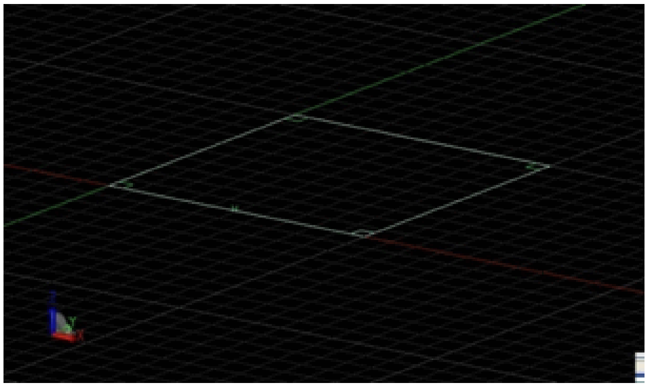
4. Click **Done**.

Editing Existing Extruded 2D Object

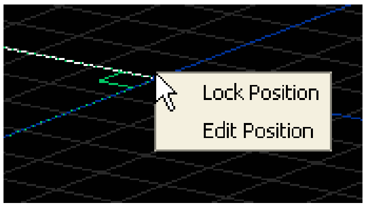
1. Click **Extrude** to open the 2D drawing space.



2. Click **Select/Manipulate** from **Tools** menu . Place the mouse over the edges or corners of rectangle, and right-click to open the **Edit/Delete** menu:

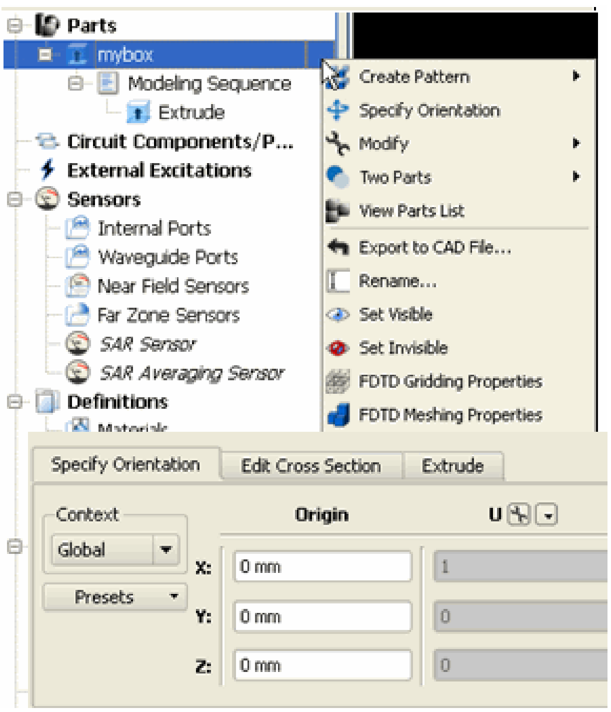


3. Delete the edges or select the vertices to edit or lock the positions.

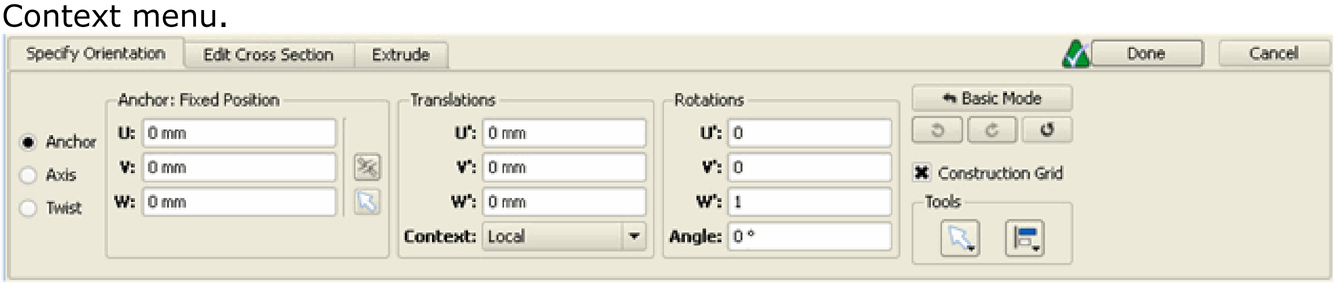


Moving (Translating)/Rotating Objects

Use "Specify Orientation" menu or tab from either the Geometry modeling menu or object created



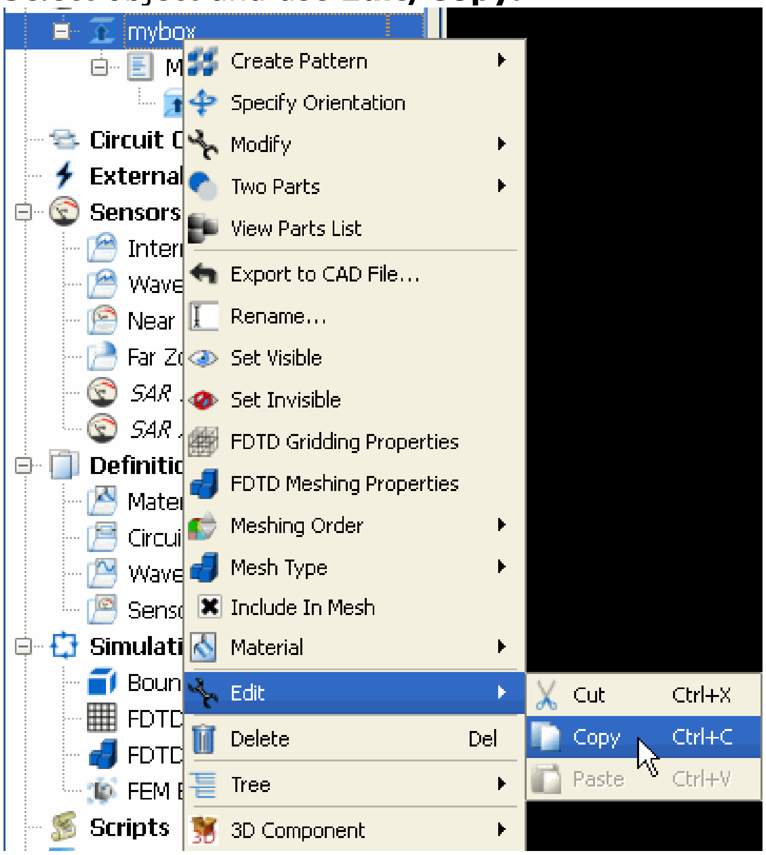
Moving objects is also referred as Translating objects in EMPro. Translation/rotation works in the local coordinate system but it can be changed to other coordinate system in the Context menu.



Copying/Deleting 2D/3D Objects

Perform the following steps for copying objects:

1. Select object and use **Edit/Copy**.



2. Select **Edit/Paste** in the **Parts** menu.

> **Note**
> Since the copy command copies onto the same position as the one being copied, you may need to translate it to other position.

Follow the same procedure for deleting an object, or you can use Delete key.

Boolean Operations

The following boolean operations are available in EMPro:

- Two Parts
- Extrude
- Revolve

The *Two Parts* tool provides several boolean operations to subtract, intersect, or unite two objects. For these operations, one object must be selected to be the *Blank*, and the other the *Tool* which acts on the blank.

Holes may also be extruded or revolved through any part with its respective tool in this menu. An object is selected in the *Pick Blank* tab and the cross section of the hole is sketched and oriented in the *Edit Profile* and *Feature Orientation* tabs, as described in the *Edit Cross Section Tab* (using) and *Specify Orientation Tab* (using), sections respectively.

Then the shape of the removed section is specified in the *Extrude Boolean* tab, or *Revolve* tab depending on which operation is selected. The *Preview* tab shows a preview of the object before the changes are formally applied to the project. For more information on defining extrusions or revolutions, refer to *3-D Solid Modeling Options* (using) in the Appendix of Geometric Modeling. An image of each boolean operation is available in *Boolean Operations* (using) in the "Appendix of Geometric Modeling".

Patterns

Patterns are created in EMPro by replicating a single selected object multiple times in one of the organized arrangements listed below.

- Linear/Rectangular
- Circular/Elliptical
  Linear/Rectangular or Circular/Elliptical patterns

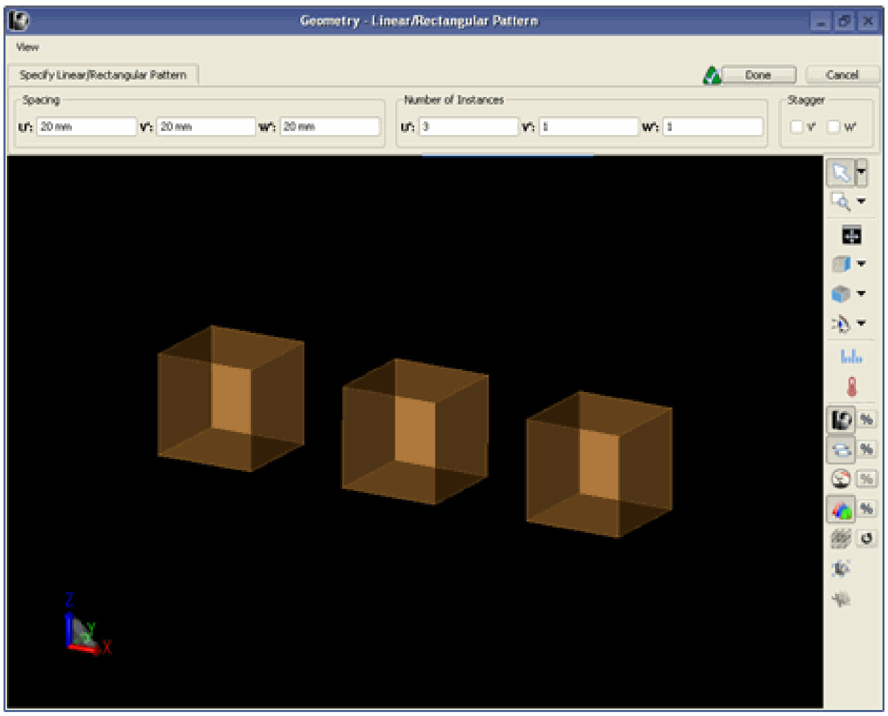


Figure: Elliptical Pattern

Adding 3D Library Components

EMPro consists of a built-in library of parameterized 3D objects that includes the following types of shapes:

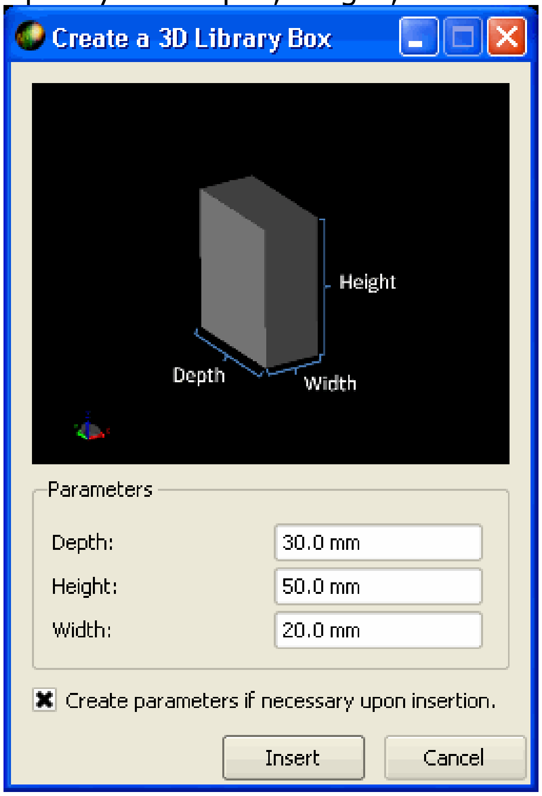
- Box
- Cylinder
- Piramid
- Ring
- Sphere
- Cone
- Helix

To insert the shape, click on the symbol at the bottom of the EMPro window. You can also edit the parameters prior to entering the object or modify them after insertion. The object can be positioned after insertion by using the **Specify Orientation** menu.

If a new parameter name is used that does was not defined in the **Parameters** menu in EMPro, the new parameter name will automatically be added.

Adding an Object from the Built-in Library

1. Select the required shape from the building blocks.
2. Specify the depth, height, and width in the Create a 3D Library Box.



3. Select the **create parameters if necessary upon insertion** checkbox.
4. Click **Insert**.

The difference of this 3D box is that the center of object is located at (0,0,0) whereas the other objects are user-definable. However, you can easily move this box to another location by translating it.

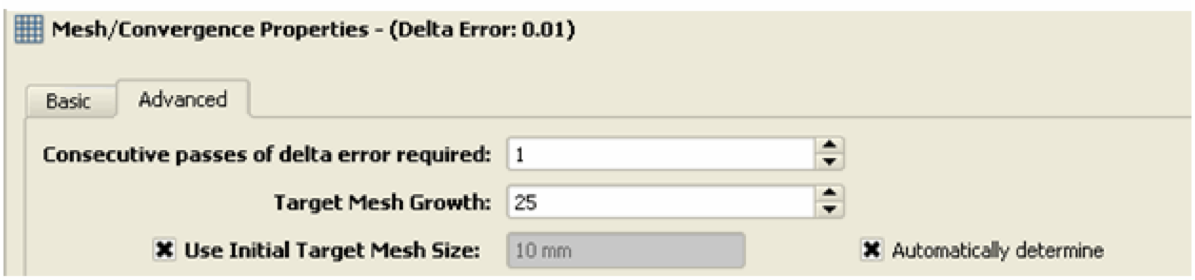
Advanced User Controlled Mesh Options

In EMPro 2010, following two advanced mesh control options are provided that allow the finer control of the initial mesh used in the adaptive refinement process:

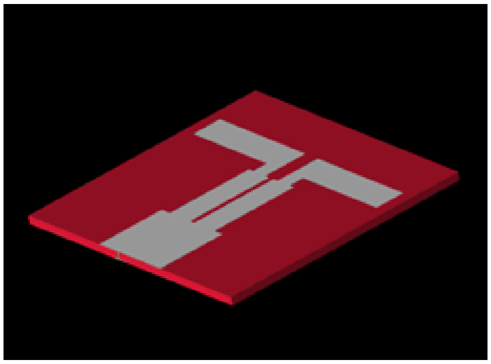
1. Initial target mesh size
2. Initial minimum mesh size

Initial Target Mesh Size

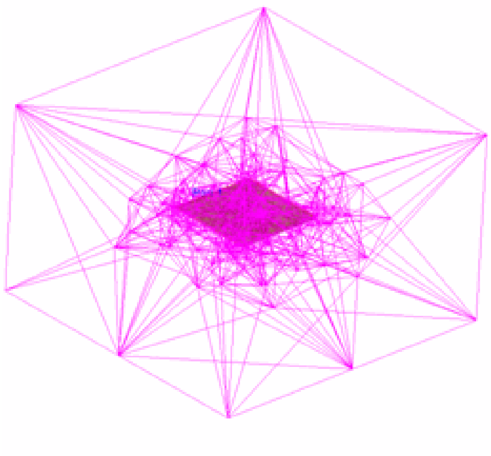
For electrically large structures it is beneficial to seed the free space with a larger number of tetrahedra than based on geometric features alone. To accomplish this, the maximum size of the edges in the initial mesh can be set to an appropriate value. Using the **Advanced** tab of the *Mesh/Convergence Properties* window (which is a part of the *Create FEM Simulation* dialog box) a length can be given (as shown in Figure below). When **automatically determine** is switched ON, EMPro provides a suggestion based on the given Frequency Plan. The automatically determined value is the wavelength in free space divided by 3.



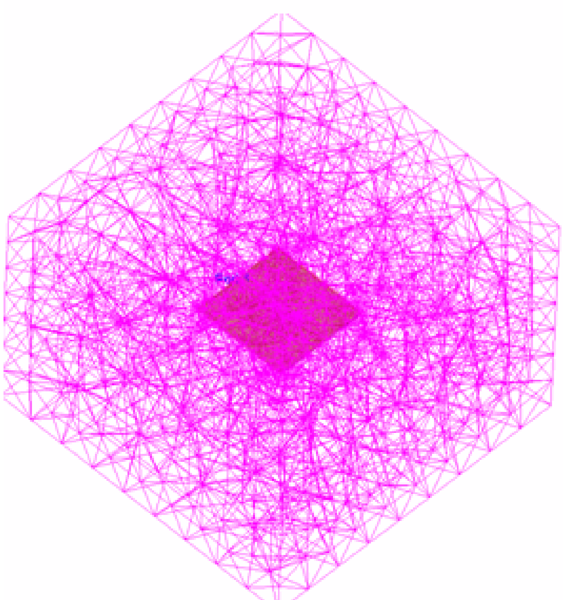
To understand this better let us take a look at the Microstrip Dipole Antenna(from **Help > Examples**):



Below is the mesh generated over freespace when **Initial Target Mesh Size** is not used:



When the same volume is meshed using the **Initial Target Mesh Size**, the mesh looks like:

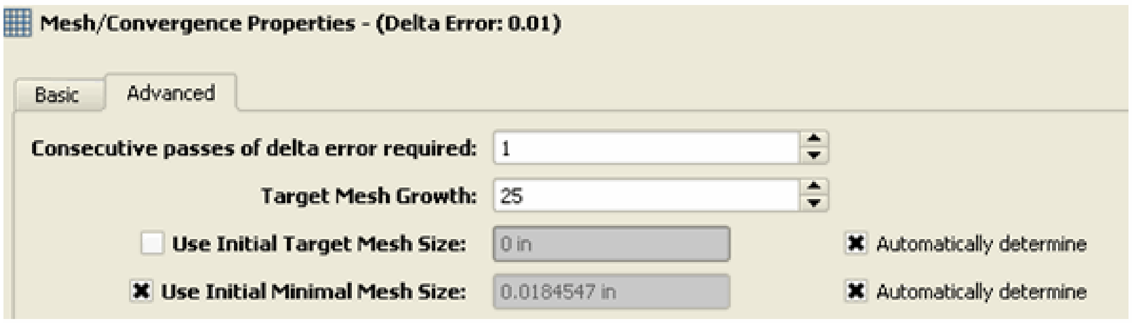


This leads to a faster convergence of iterative solver and in less number of passes in the Adaptive Frequency Sweep.

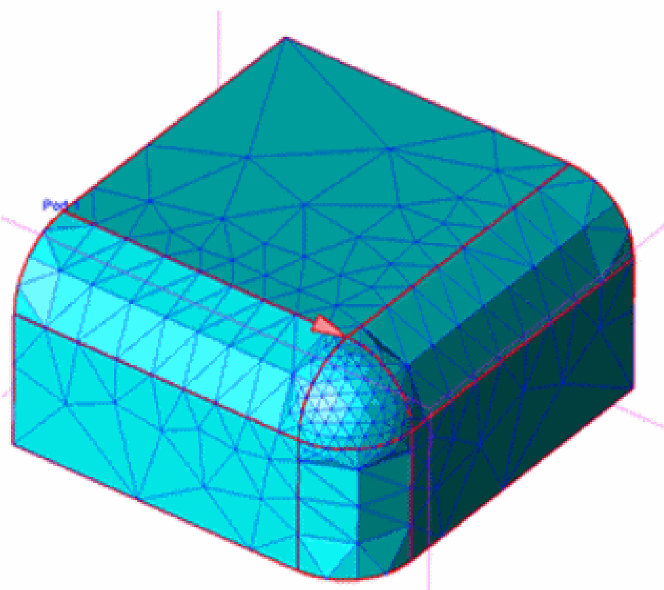
Initial Minimum Mesh Size

Initial Minimum Mesh Size controls the smallest length present in the initial mesh. It can be applied when there are geometric features present that are of less importance for the EM simulation. The result is a lower number of tetrahedra necessary to converge to the final solution. A typical use scenario for this feature is when a complex CAD model is imported or drawn where the geometric detail exceeds the detail required for the EM solution. The mesher is most effective in applying this constraint on curved surfaces.

Using the **Advanced** tab of the **Mesh/Convergence Properties** widget (which is a part of the *Create FEM Simulation* dialog box) a length can be given (as shown in Figure below).When the option **Automatically determine** is checked the EMPro GUI will provide a suggestion. The suggestion is based on the size of the geometry and the value of the **Initial Target Mesh Size**. The ratio between the **Initial Target Mesh Size** and the **Initial Minimum Mesh Size** cannot be lower than 10 to provide the mesher with enough freedom to fill the solution space with tetrahedra.



The figures below illustrate the effect on a simple example when the setting minimum size is not set.



The figures below illustrate the effect on a simple example when the setting minimum size is set to 3 mm.