



Simcenter MAGNET™ Software Tutorials

Software Version 2019.1

November 2019

**Unpublished work. © 2019 Mentor Graphics Corporation
All rights reserved.**

This document contains information that is confidential and proprietary to Mentor Graphics Corporation, Siemens Product Lifecycle Management Software Inc., and their affiliates (collectively, "Siemens"). The original recipient of this document may duplicate this document in whole or in part for internal business purposes only, provided that this entire notice appears in all copies. In duplicating any part of this document, the recipient agrees to make every reasonable effort to prevent the unauthorized use and distribution of the proprietary information.

This document is for information and instruction purposes. Siemens reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Siemens to determine whether any changes have been made.

The terms and conditions governing the sale and licensing of Siemens products are set forth in written agreements between Siemens and its customers. No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Siemens whatsoever.

SIEMENS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

SIEMENS SHALL NOT BE LIABLE FOR ANY INCIDENTAL, DIRECT, INDIRECT, SPECIAL, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING BUT NOT LIMITED TO LOST PROFITS) ARISING OUT OF OR RELATED TO THIS PUBLICATION OR THE INFORMATION CONTAINED IN IT, EVEN IF SIEMENS HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

U.S. GOVERNMENT LICENSE RIGHTS: The software and documentation were developed entirely at private expense and are commercial computer software and commercial computer software documentation within the meaning of the applicable acquisition regulations. Accordingly, pursuant to FAR 48 CFR 12.212 and DFARS 48 CFR 227.7202, use, duplication and disclosure by or for the U.S. Government or a U.S. Government subcontractor is subject solely to the terms and conditions set forth in the license agreement provided with the software, except for provisions which are contrary to applicable mandatory federal laws.

TRADEMARKS: The trademarks, logos and service marks ("Marks") used herein are the property of Siemens or other parties. No one is permitted to use these Marks without the prior written consent of Siemens or the owner of the Mark, as applicable. The use herein of a third-party Mark is not an attempt to indicate Siemens as a source of a product, but is intended to indicate a product from, or associated with, a particular third party.

The registered trademark Linux® is used pursuant to a sublicense from LMI, the exclusive licensee of Linus Torvalds, owner of the mark on a world-wide basis.

Introduction to the tutorials

This document includes eleven tutorials to help you increase your skills with Simcenter MAGNET. Each tutorial teaches you basic procedures that you can apply to your own models.

2D Tutorials

- Time-harmonic solution: Cylindrical Shield
- Transient solution: Felix long cylinder
- Basic model tutorial: Spherical Shield
- Magnetostatic solution: Spherical Shield
- Time-harmonic solution: Spherical Shield
- Transient 2D with Motion solution - TEAM Problem 30

3D Tutorials

- Magnetostatic solution: Pot-core with a coil
- Time-harmonic solution: Bath plate
- Transient solution: Felix short cylinder
- Parameterization tutorial: C-core with a rotating block
- Transient 3D with Motion solution - Permanent magnet stepper motor

Additional information

The Getting Started Guide introduces you to the basic Simcenter MAGNET concepts. More information on the procedures and concepts of model building with Simcenter MAGNET is found in the Help, included with each package.

Features showcase

Tutorial #1

2D Time-harmonic solution -- Cylindrical Shield (Translational Geometry)

Features you will learn:

- Changing the material of a component
- Refining the mesh by changing the maximum element size
- Editing the properties of coils
- Setting the source frequency
- Setting the linear solving option
- Viewing the time-averaged ohmic loss in the conductor
- Viewing the contour plot of the model
- Probing a field value

Tutorial #2

2D Transient solution -- Felix long cylinder (Translational Geometry)

In this tutorial, the cylindrical shield model built in Tutorial #1 is used as the basis for the Felix long cylinder problem.

Features you will learn:

- Changing the material of a component
- Editing the properties of a coil
- Creating a circuit
- Defining an exponential waveform
- Viewing a contour plot
- Animating a contour plot
- Viewing the instantaneous ohmic loss of each conducting component
- Graphing the power across time

Tutorial #3

2D basic model -- Spherical shield (Rotational Geometry)

In this tutorial, a spherical shield model is built with the Simcenter MAGNET defaults.

Features you will learn:

- Drawing with the Keyboard Input bar
- Creating components in a rotational direction
- Rotating the display of a model
- Refining the mesh by changing the maximum element size
- Defining boundary conditions
- Creating a coil

Tutorial #4

2D Magnetostatic solution -- Spherical shield (Rotational Geometry)

In this tutorial, the spherical shield model built in Tutorial #3 is edited for a magnetostatic solution.

Features you will learn:

- Changing the material of a component
- Editing the properties of a coil
- Solving with the default options
- Viewing the contour plot of the model
- Probing a field value

Tutorial #5

2D Time-harmonic solution -- Spherical shield (Rotational Geometry)

In this tutorial, the spherical shield model built in Tutorial #3 is edited for a time-harmonic solution .

Features you will learn:

- Changing the material of a component
- Editing the properties of a coil
- Setting the source frequency
- Setting the linear solving option
- Viewing the time-averaged ohmic loss in the conductor
- Probing a field value
- Viewing the contour plot of the model

Tutorial #6

2D Transient with Motion solution – TEAM Problem 30

In this tutorial, steps guide you through creating a half-model of a 3-phase induction machine and then analyzing the device in Simcenter MAGNET, which includes setting up and analyzing problems for a locked rotor analysis, determining the torque at a speed of 200 rad/s, and examining rotor speed and torque during start-up conditions.

Features you will learn:

- Setting up the work environment by modifying initial settings and the viewing area
- Building the geometric model using the Keyboard Input Bar
- Transforming construction slice edges
- Creating new user materials
- Setting up the problem, which consists of making components and coils, assigning boundary conditions, modifying the mesh, and creating circuits
- Making a motion component
- Generating the time-harmonic and transient field solutions using Simcenter MAGNET's Time-Harmonic 2D and Transient 2D with Motion solvers
- Analyzing the results, which includes:
 - viewing the torque of the locked rotor and viewing its contour plot
 - graphing the torque for the rotor when it is rotating at 200 rad/s
 - graphing the speed and torque of the rotor during the start-up phase

Tutorial #7

3D Magnetostatic tutorial -- Pot-core with a coil

In this tutorial, you will create a pot-core model with a coil. You will also refine the initial (default) mesh.

Features you will learn:

- Distorting the shape of a component
- Building a coil
- Refining the mesh by changing the maximum element size
- Viewing an arrow plot

Tutorial #8

3D Time-harmonic tutorial -- Bath plate

In this tutorial, you will create a conducting ladder with two holes (the Bath plate TEAM problem).

Features you will learn:

- Creating a current-driven coil
- Modeling a closed conducting loop using a voltage-driven coil
- Setting the source frequency
- Creating a contour
- Using the Calculator to get the magnitude of B_z
- Graphing the magnitude of B_z along the contour

Tutorial #9

3D Transient tutorial -- Felix short cylinder

In this tutorial, you will build a cylindrical shield and solve it using Magnet's 3D Transient Solver.

Features you will learn:

- Creating a new material
- Creating a circuit
- Defining an exponential waveform
- Viewing a shaded plot
- Animating a shaded plot
- Viewing the stored energy in the system
- Graphing the energy across time

Tutorial #10

Parameterization tutorial -- C-core with a rotating block

In this tutorial, you will parameterize the position of a permanent magnet and view post processing results across multiple solutions.

Features you will learn:

- Creating a user-defined parameter
- Rotating the geometric position of a component using parameterization
- Viewing instantiations of a model
- Viewing force and torque on a body
- Graphing the Z component of torque over multiple solutions
- Creating an animation of the shaded plot over multiple solutions

Tutorial #11

3D Transient with Motion solution – Permanent magnet stepper motor

The stepper motor used in this tutorial is made of a stator with eccentric pole faces and a rotor made of samarium cobalt permanent magnet that is magnetized in a fixed direction. The rotor rotates in steps of 180 degrees. Each step in the rotor is due to a short pulse from a current source. To obtain a unidirectional motion, the current pulse is alternating. In this tutorial, the response due to one pulse is examined.

Features you will learn:

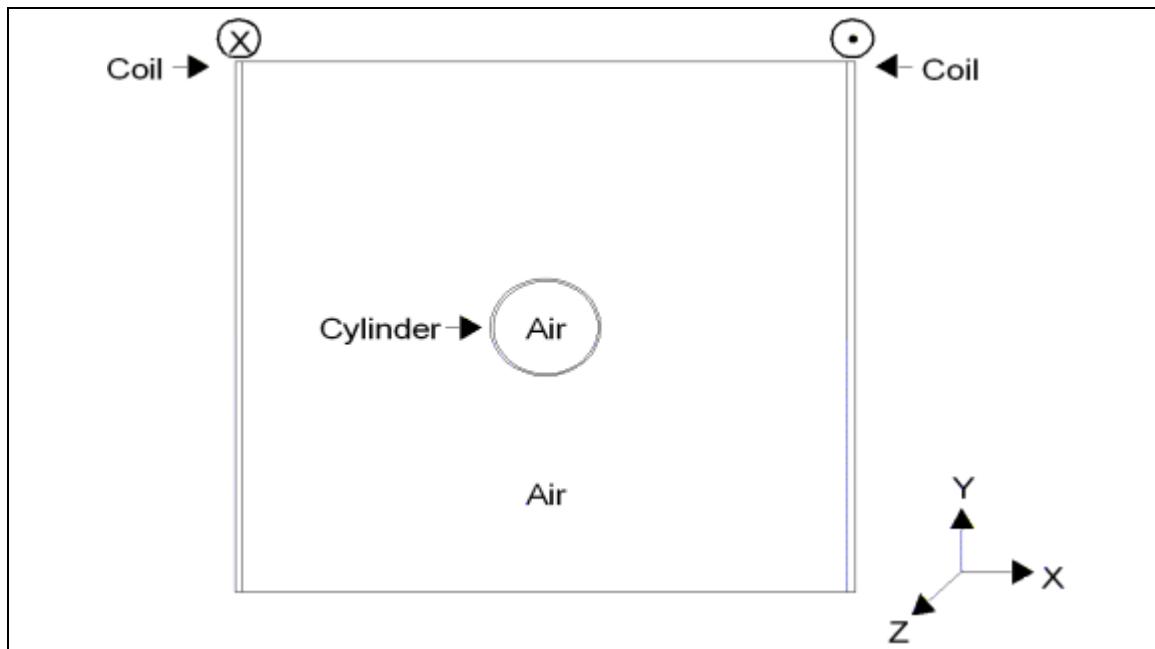
- Setting up the work environment by modifying initial settings and the viewing area.
- Building the geometric model using the Keyboard Input Bar.
- Setting up the problem -- this consists of making components and coils, and modifying the mesh.
- Making a motion component along with a Remesh Region component.
- Generating the transient field solution using Simcenter MAGNET's Transient 3D with Motion solver.
- Analyzing the results, which includes:
 - graphing the position, magnetic torque, velocity, and acceleration of the motion component
 - creating an animation of the shaded plot of $|B|$.

Tutorial #1

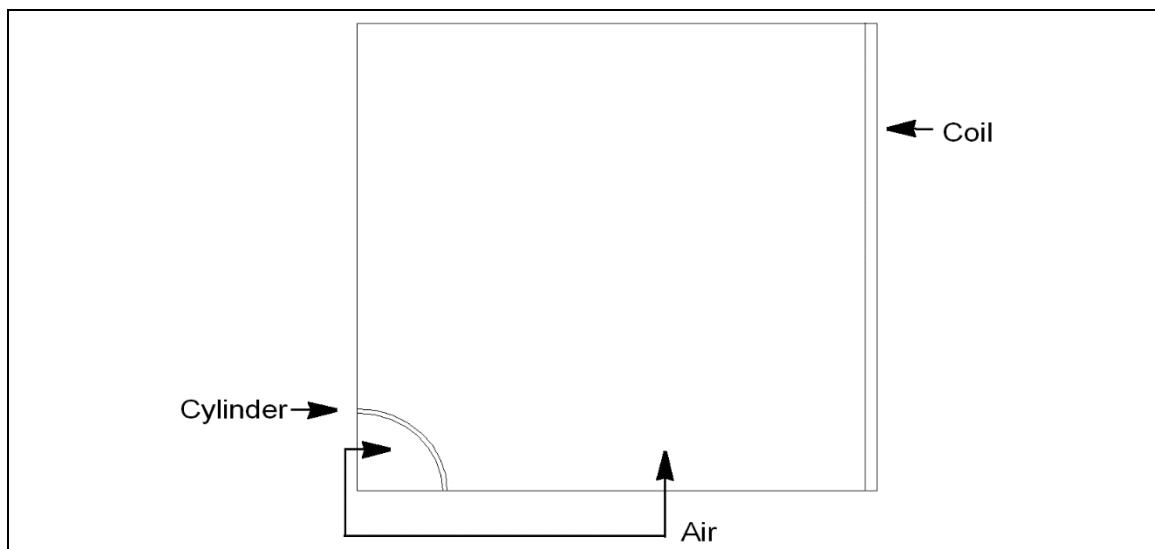
*2D Time-harmonic
Cylindrical shield*

1 Modeling plan

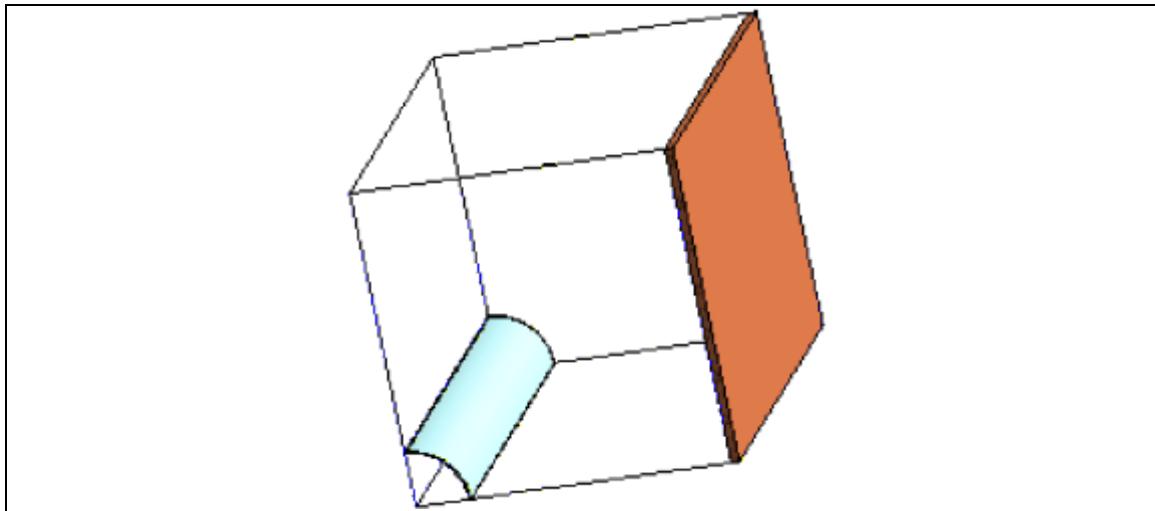
The problem is comprised of a ferromagnetic hollow cylinder, infinitely long in the z-direction, lying in a uniform field. A stranded coil of infinite length, lying on either side of the cylinder, provides the uniform field.



Symmetry conditions allow for only one-quarter of the problem to be modeled. The model is built from three components: a cylinder, a coil, and an air space that encompasses the two other components. The outline of the model is shown below.

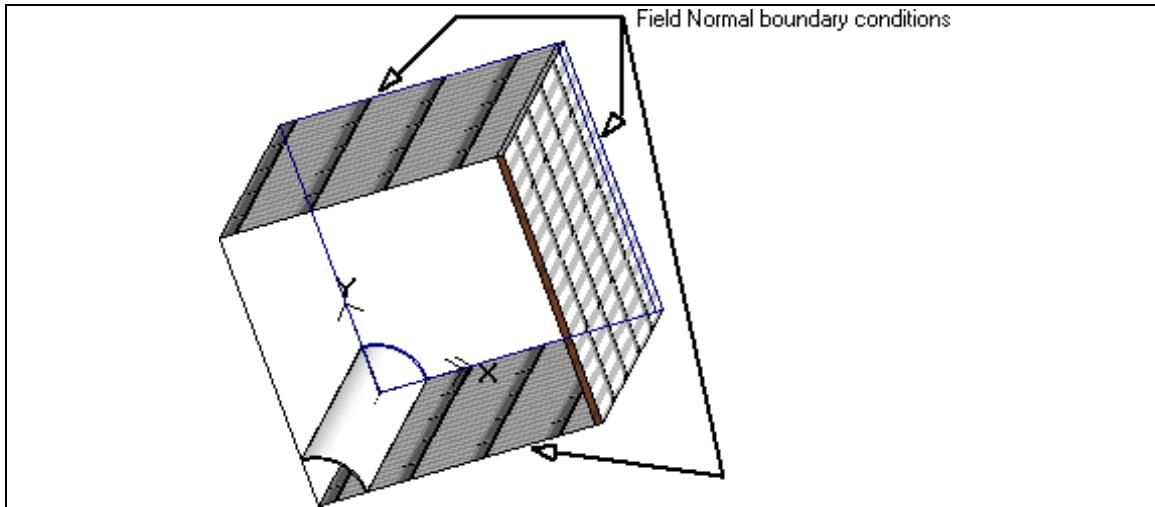


After it is drawn, the outline is swept into components and two coils are created from the solid components.

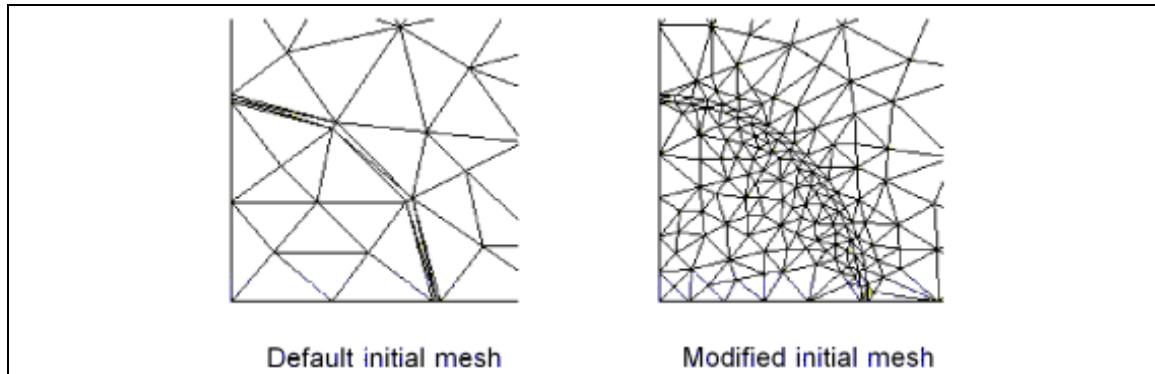


Boundary conditions are applied to indicate symmetry. The Field Normal boundary condition is applied to the top, bottom, and right boundaries of the model. The default boundary condition, Flux Tangential, is automatically applied to the remaining boundaries.

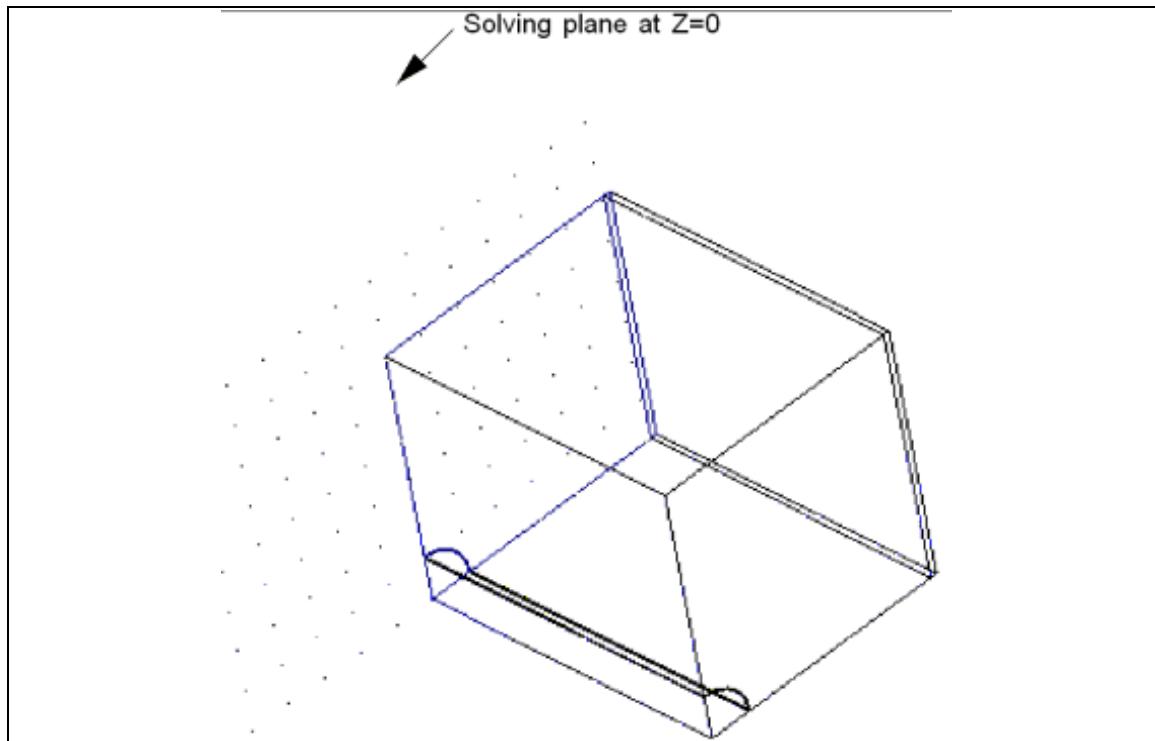
Tip The Field Normal boundary condition on the right boundary of the model (representing the outside of the coil) forces the coil flux outside to infinity. If the surface had the Flux Tangential boundary condition, the coil flux would be forced to return inside the coil, which would give incorrect field values inside the coil.



The density of the mesh will be increased in the area of the cylinder to improve solution accuracy.



The model is solved at the XY plane where Z=0 (the default position of the construction slice).



2 Open a new model

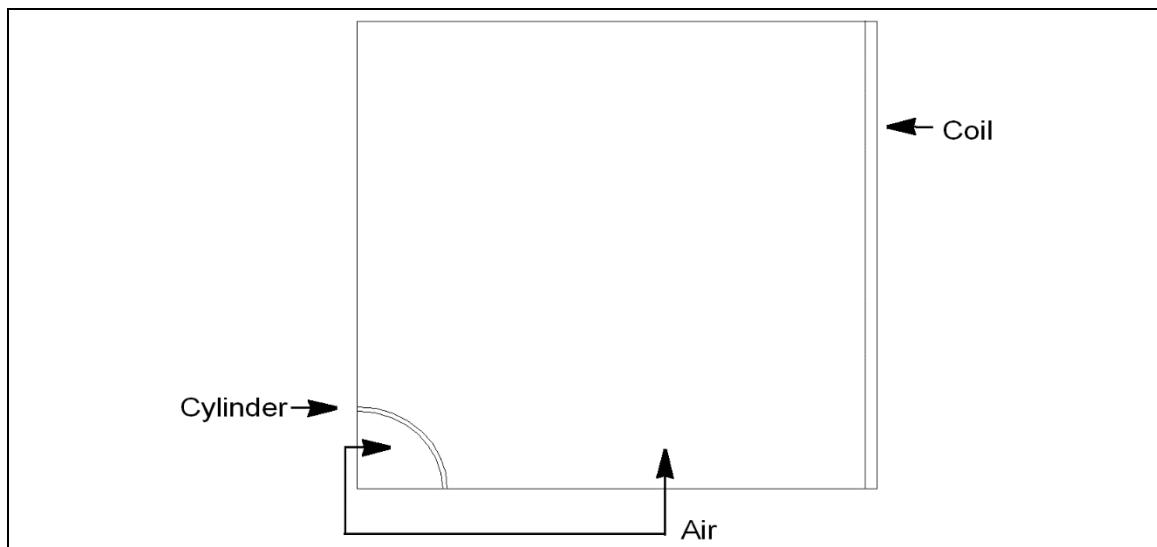
1. Start Simcenter MAGNET.
The Main window appears.
2. If Simcenter MAGNET was already running, select New from the File menu to open a new model.
If you have already used Simcenter MAGNET, the window settings are those that were last active. To maximize the window, click  on its top right corner.

3 Name the model

1. On the File menu, click *Save* or *Save As*.
2. In the *Save As* dialog box, type **Cylindrical Shield - Time-Harmonic** as the name of the model.
3. Choose the drive and directory where you want to place the model.
4. Click *Save As*.

4 Build the geometric model

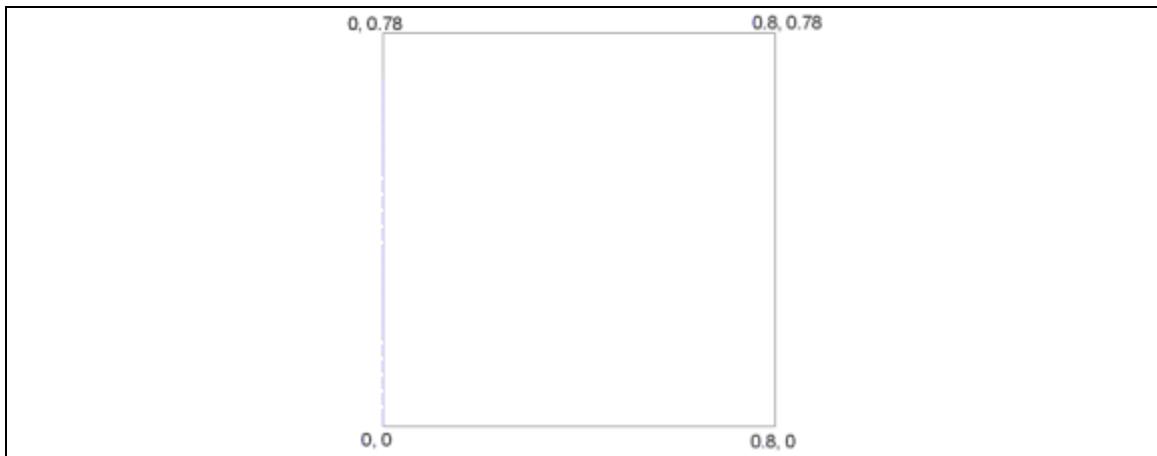
The outline of the model is shown below.



5 Create the air space component

5.1 Draw the outline of the air space

The outline of the air space is shown in the diagram below. Dimensions are in meters, which is the default drawing unit in Simcenter MAGNET.



1. If the Keyboard Input bar is not already displayed at the bottom of the Main window, select *Keyboard Input Bar* on the Tools menu.



2. See that the Keyboard Input bar is set to (Cartesian) and (Absolute).
3. On the Draw toolbar, click .
4. On the View toolbar, click .
- This option updates the display of the model to fit inside the View window.
5. In the Keyboard Input bar, enter the following coordinates to draw the air space.

Start coordinates 0, 0 Press ENTER

End coordinates 0.8, 0 Press ENTER

End coordinates 0.8, 0.78 Press ENTER

End coordinates 0, 0.78 Press ENTER

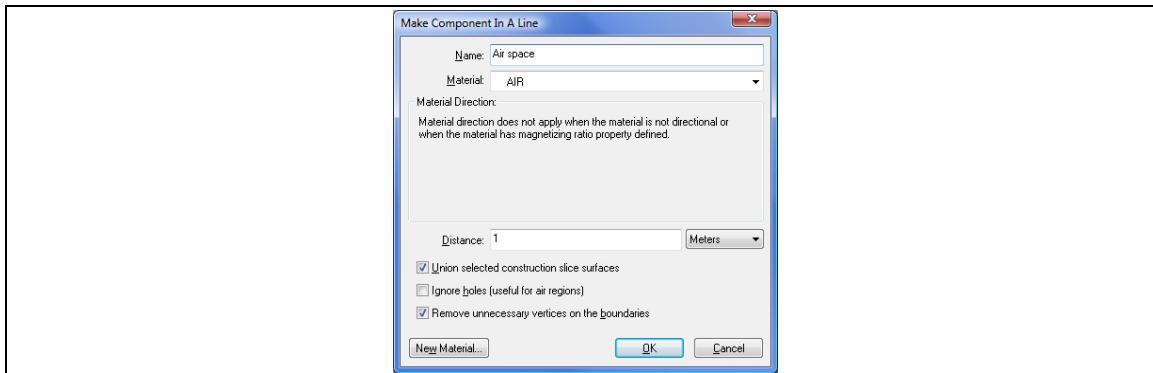
End coordinates 0, 0 Press ENTER

6. Press ESC to stop drawing.

5.2 Make the component of the air space

A component can now be made from the surface that you have drawn. The surface is swept over a length of one meter. Components are created using the Make Component dialog box.

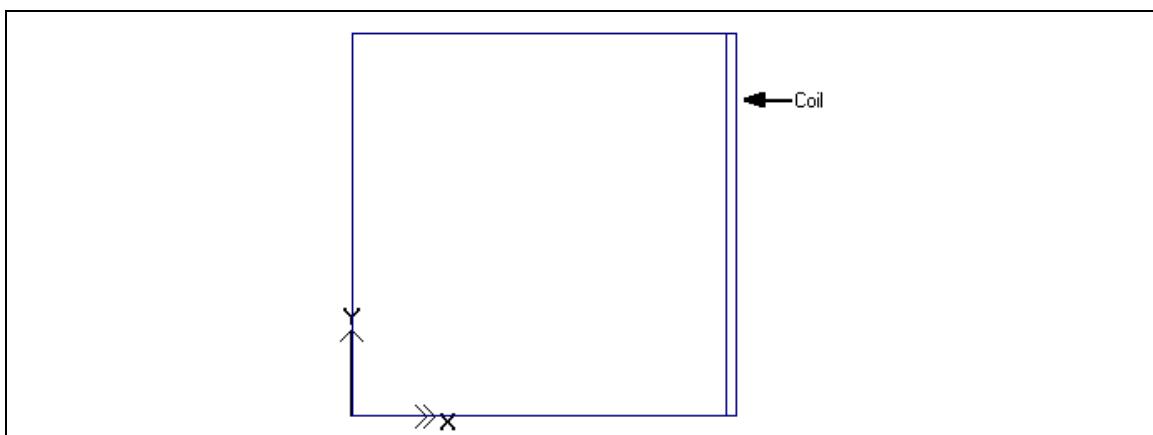
1. On the Edit menu, click Select Construction Slice Surfaces.
2. Click the mouse pointer inside the surface of the air space.
The surface is highlighted when selected.
3. On the Model toolbar, click  (Make Component in a Line tool).
The *Make Component In A Line* dialog box appears.



4. In the *Name* box, type **Air space**.
5. In the *Material* drop-down list, make sure AIR is selected.
6. In the *Distance* box, type **1**.
7. Click OK to accept the settings.
The component is created.
8. On the File menu, click *Save*.

6 Create the coil component

The outline of the coil component is shown below.



6.1 Draw the outline of the coil

1. On the Draw toolbar, click  (Line drawing tool).
2. In the Keyboard Input bar, enter the following coordinates to complete the drawing of the coil.

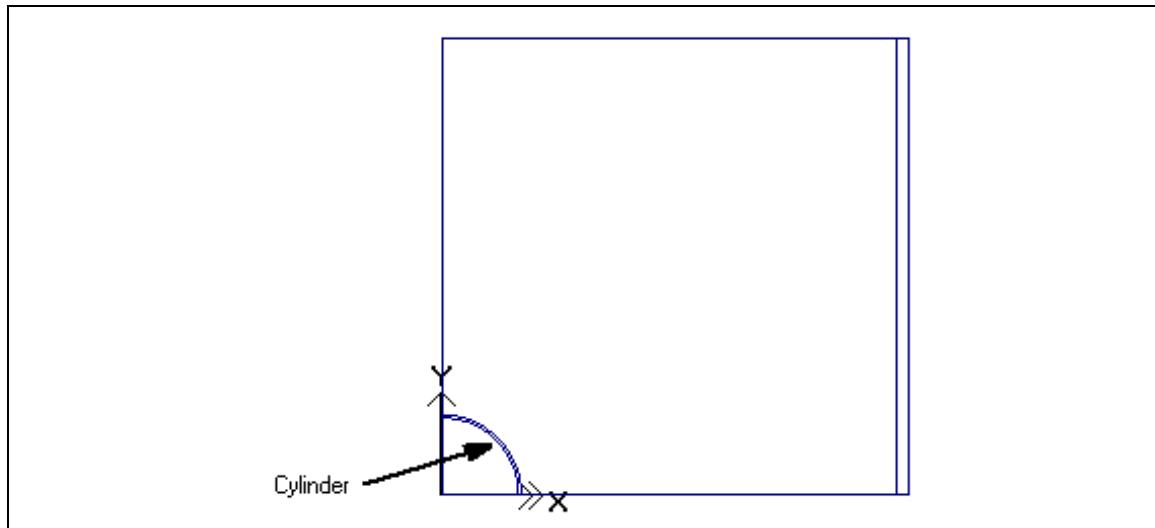
Start coordinates	0.78, 0	Press ENTER
End coordinates	0.78, 0.78	Press ENTER
3. Press ESC to stop drawing.

6.2 Make the component of the coil

1. On the Edit menu, click Select Construction Slice Surfaces.
2. Click the mouse pointer inside the surface of the coil.
3. On the Model toolbar, click  (Make Component in a Line tool).
4. In the *Name* box, type **Coil component**.
5. In the *Material* drop-down list, select **Copper: 5.77e7 Siemens/meter**.
6. Click OK to accept the settings.
The component is created.
7. On the File menu, click *Save*.

7 Create the cylinder component

The cylinder is positioned at the lower left corner of the air space. It has a wall thickness of .0048 meter and an inner radius of 0.1317 meter.



7.1 Draw the outline of the cylinder

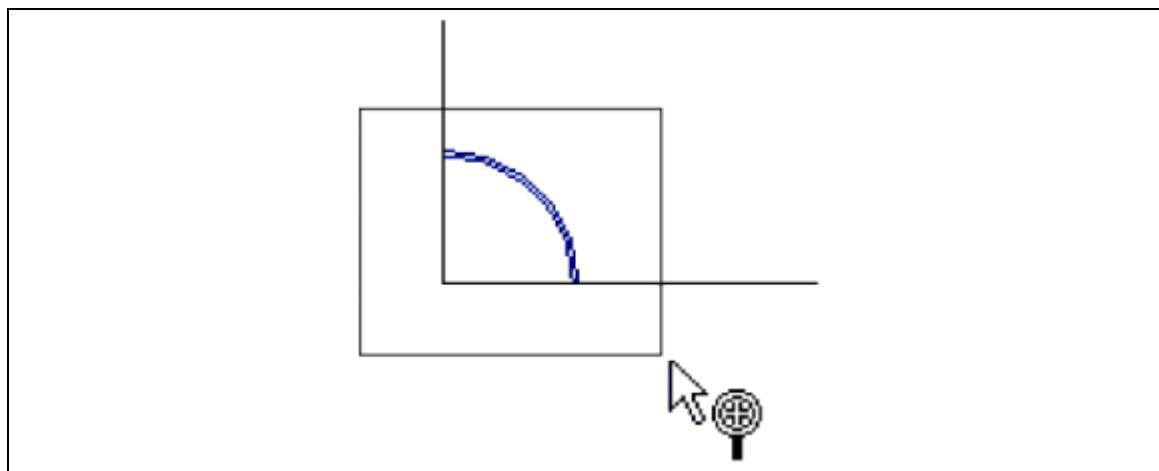
1. On the Draw toolbar, click  (Center, Start, End).
2. In the Keyboard Input bar, enter the following coordinates for the inner and outer arcs of the cylinder, respectively.

Note Arcs are drawn in a counter-clockwise direction.

Center coordinates	0, 0	Press ENTER
Start coordinates	0.1317, 0	Press ENTER
End coordinates	0, 0.1317	Press ENTER
Center coordinates	0, 0	Press ENTER
Start coordinates	0.1365, 0	Press ENTER
End coordinates	0, 0.1365	Press ENTER

7.2 Make the component of the cylinder

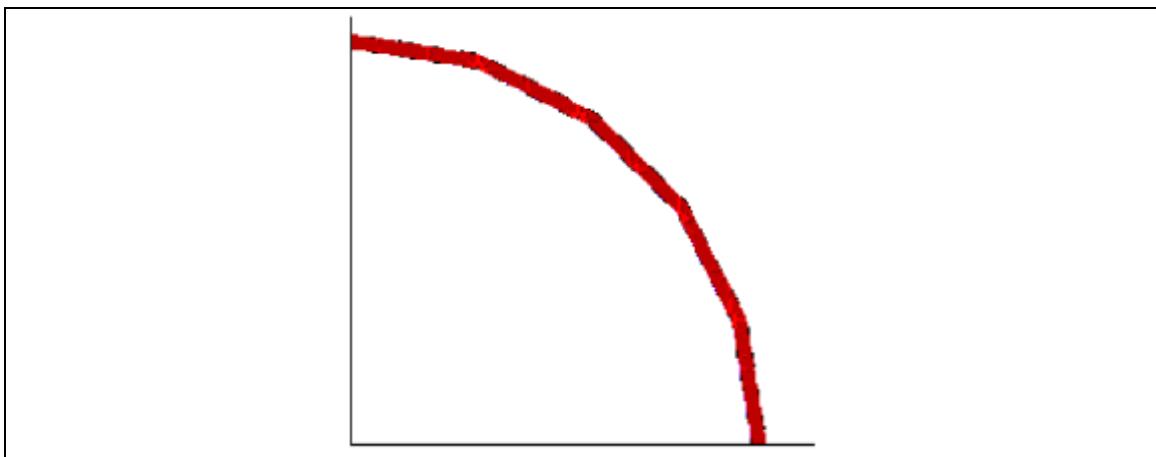
1. On the View toolbar, click  (Examine Model).
2. Hold down the CTRL key and the left mouse button to form a rectangular box around the cylinder.



3. Release the mouse pointer.
The area enclosed by the rectangle is enlarged.
4. On the Edit menu, click Select Construction Slice Surfaces.

5. Click the mouse pointer inside the surface of the cylinder.

The surface is highlighted when selected.



6. On the Model toolbar, click  (Make Component in a Line tool).
7. In the Name box, type **Cylinder**.
8. Click the *New Material* button located at the bottom left corner of the dialog.
For this problem, you will have to create a new material in your material database.
9. On the General page, specify the following:
 - Name: Aluminum 6061
 - Display color: *Click Display Color and select an appropriate color*
 - Transparency: *Optional*
 - Description: *Optional*
 - Categories: *Optional*
10. Click *Next*.
11. On the Options page, select the following:
 - Magnetic Permeability
 - Electric Conductivity
12. Using the *Next* button to advance to the appropriate pages, enter the following values:
 - Temperature (Celsius) = 20 Press ENTER
 - Relative Permeability = 1 Press ENTER
 - Coercivity (Amps/m) = 0 Press ENTER
 - Temperature (Celsius) = 20 Press ENTER
 - Conductivity (Siemens/m) = 2.538e7 Press ENTER
13. Once you have entered all the values, advance to the Confirmation page and click *Finish* to create the new material.
14. In the *Material* drop-down list, verify that **Aluminum 6061** is selected.
15. Click *OK* to accept the settings and create the component.
16. On the File menu, click *Save*.

7.3 Delete the lines on the construction slice

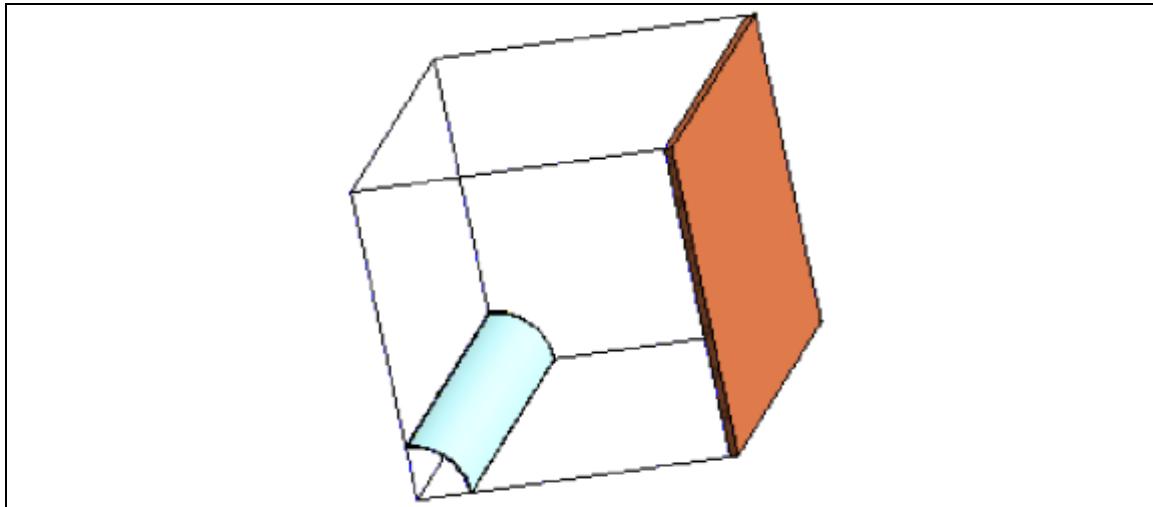
We no longer need the construction slice lines that were used to create the components, so we will proceed to remove them.

1. On the Edit menu, click Select Construction Slice Edges.
2. On the Edit menu, click *Select All*.
3. On the Edit menu, click *Delete*.

7.4 Rotate the display of the model

1. On the View toolbar, click  (Automatic View All).
2. On the View toolbar, click  (Examine Model).
3. Holding down the left mouse button, drag the mouse pointer in one or more of the following directions:
 - Drag up to rotate the display upward.
 - Drag down to rotate the display downward.
 - Drag right to rotate the display toward the right.
 - Drag left to rotate the display toward the left.
4. Release the mouse button.

The display is rotated about the center of the model.



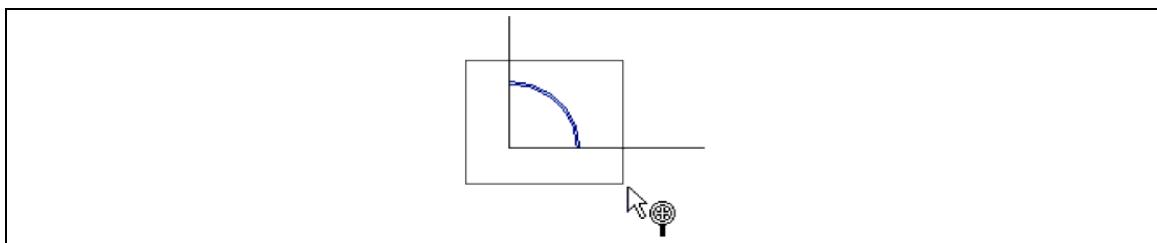
8 Modify the mesh

In the 2D finite element method of analysis, the solution domain is divided into a mesh of triangular elements. The accuracy of the solution depends upon the nature of the field and the size of the mesh elements. In regions where the direction or magnitude of the field is changing rapidly, high accuracy requires smaller elements. One method of increasing the mesh density is to set the *maximum element size* for a component volume or specific faces of a component. The following procedures will demonstrate this method.

8.1 View the initial mesh

Before changing the *maximum element size*, the default initial mesh can be viewed.

1. On the Preset View toolbar, click  [Show XY (+Z)].
2. With the Examine Model tool still active, hold down the CTRL key and the left mouse button to form a rectangular box around the cylinder.



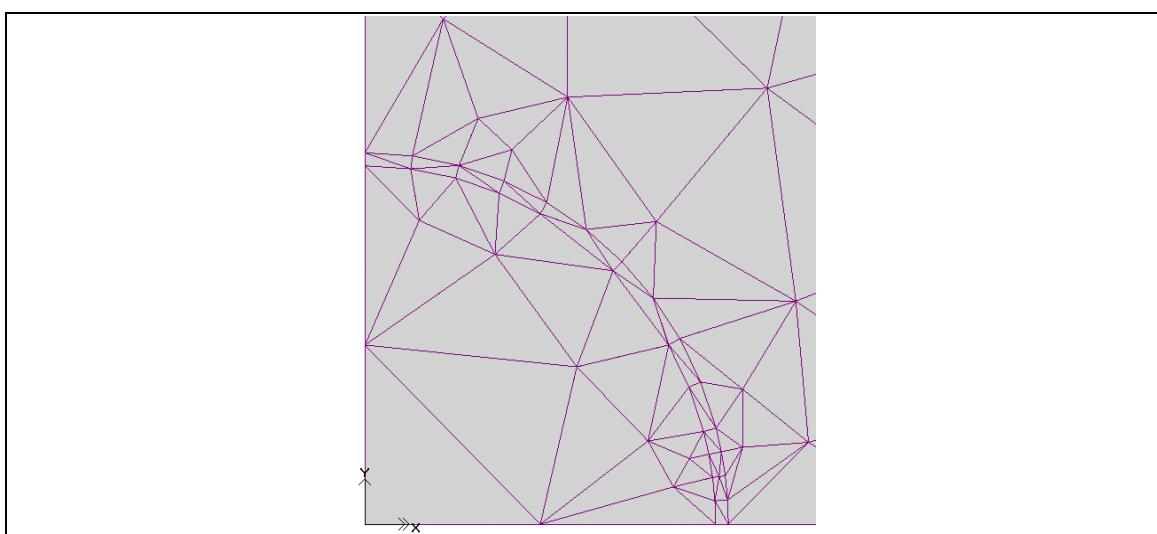
3. Release the mouse pointer.

The area enclosed by the rectangle is enlarged.

4. On the View menu, click *Initial 2D Mesh*.

The initial mesh appears in the View window. It is displayed on the XY plane, at Z=0.

The mesh should look like the following diagram.

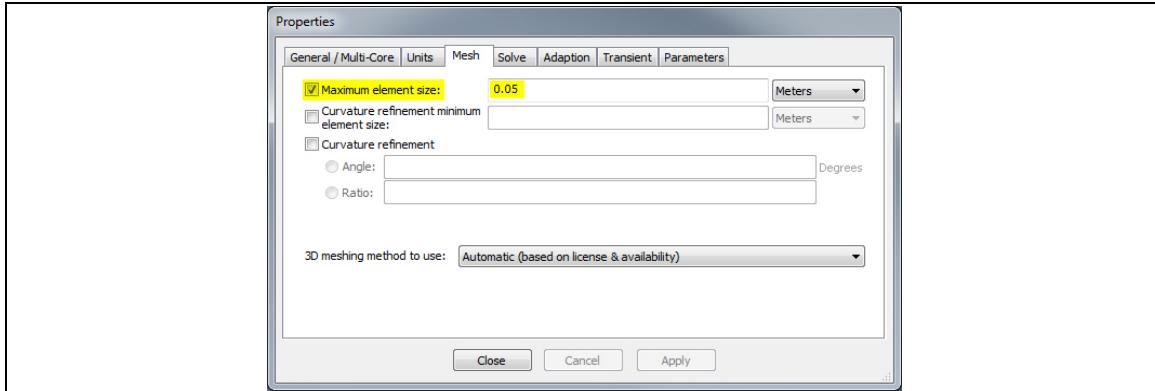


5. On the View menu, click *Solid Model*.

8.2 Set the maximum element size

For this procedure, we will set the maximum element size for the model, a component (the Cylinder), and some vertices of that *Cylinder* component.

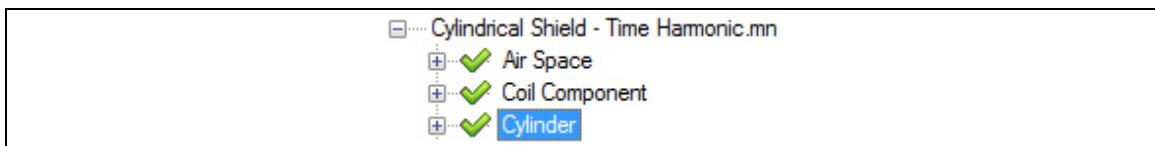
1. In the Object page of the Project bar, select *Cylindrical Shield – Time harmonic.mn*.
2. On the Edit menu, click *Properties*.
3. In the Properties dialog, select the *Mesh* tab.
4. Click inside the *Maximum element size* checkbox, and then type **0.05** in the text box.



5. Click Apply.

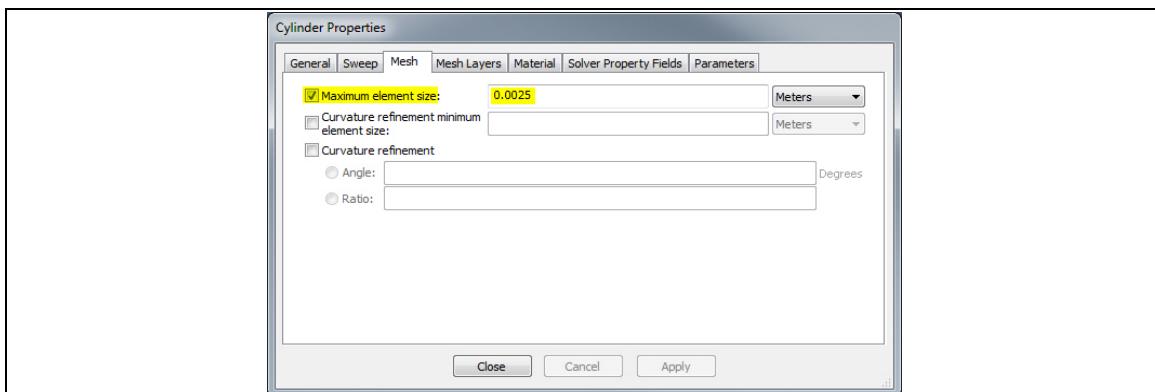
Tip Clicking Apply, instead of OK, keeps the dialog open and allows us to proceed to the next object without having to repeat step 2.

6. In the Object page, select the *Cylinder* component.



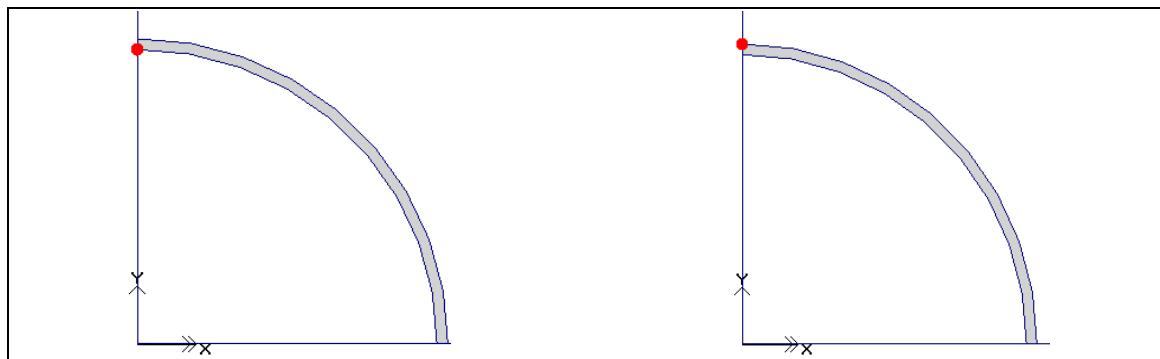
The Properties dialog is retitled *Cylinder Properties*.

7. Change the 0.05 previously set to **0.0025** in the text box.

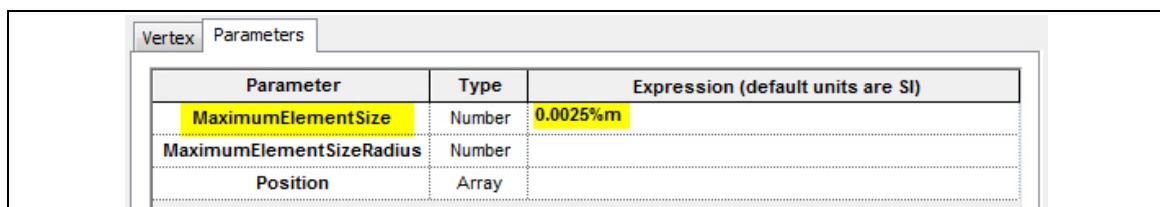


8. Click Apply.

9. On the Edit menu, click *Select Component Vertices*.
10. Click the first vertex as shown below on the left.



- The Properties dialog is retitled *Properties*.
11. Select the *Parameters* tab.

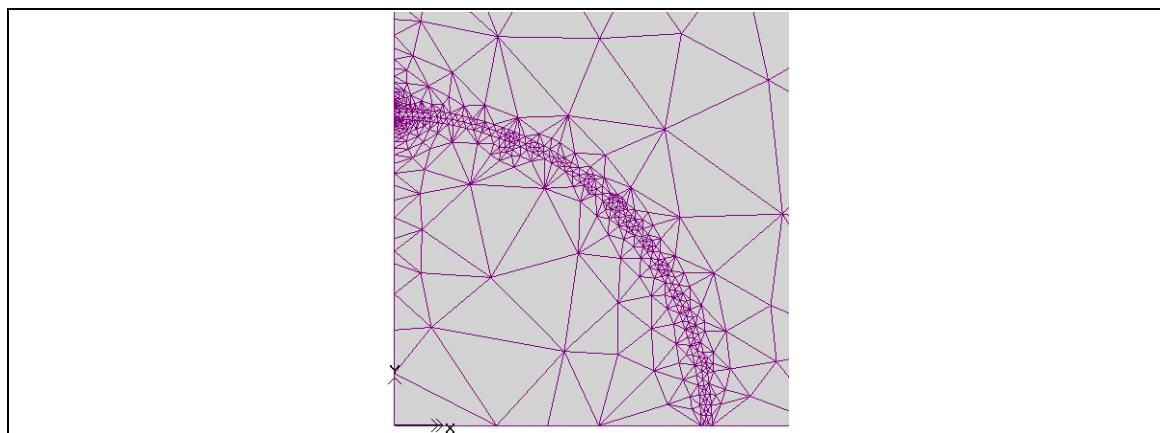


12. In the *Expression* column of the *MaximumElementSize* parameter, change the 0.0025% m previously set to **0.0005% m** .
13. Press Tab, and then click *Apply*.
14. Click the second vertex as shown above on the right.
15. As before, change the 0.0025% m previously set to **0.0005% m** .
16. Click *Close*.

8.3 View the changes to the mesh

1. On the View menu, click *Initial 2D Mesh*.

The mesh updates and should look like the following diagram.



2. On the View menu, click *Solid Model*.

9 Define boundary conditions

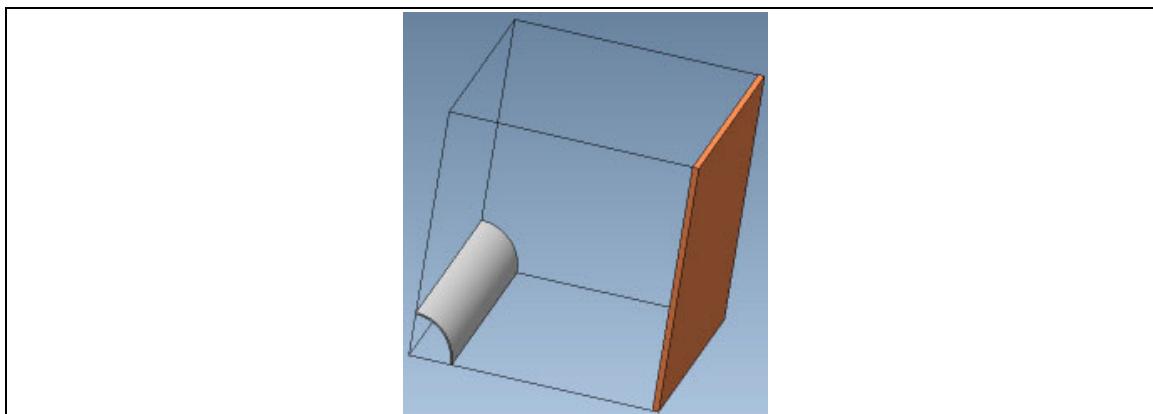
The Field Normal boundary condition is applied to three faces of the air space component: the top, bottom, and right faces. The default boundary condition, Flux Tangential, is automatically applied to the remaining faces.

The Field Normal boundary condition constrains to zero the tangential component of the magnetic field. The field is made normal (perpendicular) to the boundary. The Flux Tangential boundary condition constrains to zero the normal component of the magnetic flux density. The flux is made to flow tangential to (alongside) the boundary.

9.1 Apply the Field Normal boundary condition

The Object Page of the Project bar lists all of the objects of the model. You can select objects using the Object page.

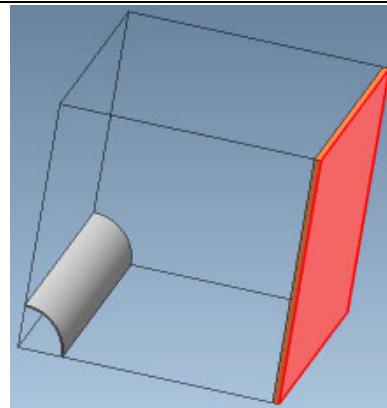
1. On the View toolbar, click  (Automatic View All).
2. On the View toolbar, click  (Examine Model).
3. Holding down the left mouse button, drag the mouse pointer to rotate the model to a 3D view (similar to the diagram below). This rotation will display the surfaces to which the Field Normal boundary condition will be applied.



4. In the Object page, click the plus sign (+) beside Air space.
The faces of the component are listed.
5. Click *Face#4*.

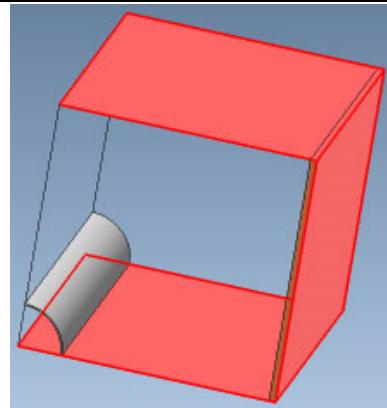


The face is highlighted in the View window.



6. While holding down the CTRL key, also select *Face#3* and *Face#5* of the *Air space* component.

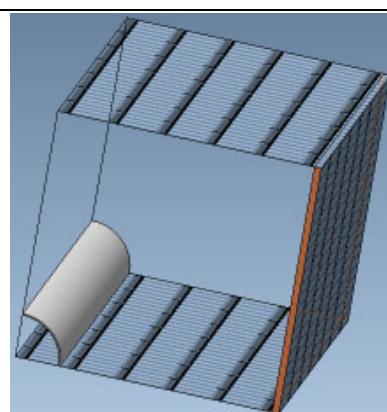
The three faces (*Face#3*, *Face#4*, and *Face#5*) are highlighted in the View window.



Tip You can select multiple objects by holding down the SHIFT or CTRL keys while clicking on additional objects. While holding down CTRL, you can click on an object a second time to de-select it.

7. On the Boundary Condition toolbar, click (Field Normal).

The Field Normal boundary condition is applied to the selected surfaces.



10 Create the coils

The coils are created from the Coil component and the Cylinder. After the coils are created, the default properties are edited.

1. On the Object page, select *Coil component*.
2. On the Model menu, click *Make Simple Coil*.
The coil is listed in the Object page as Coil#1.
3. On the Object page, select *Cylinder*.

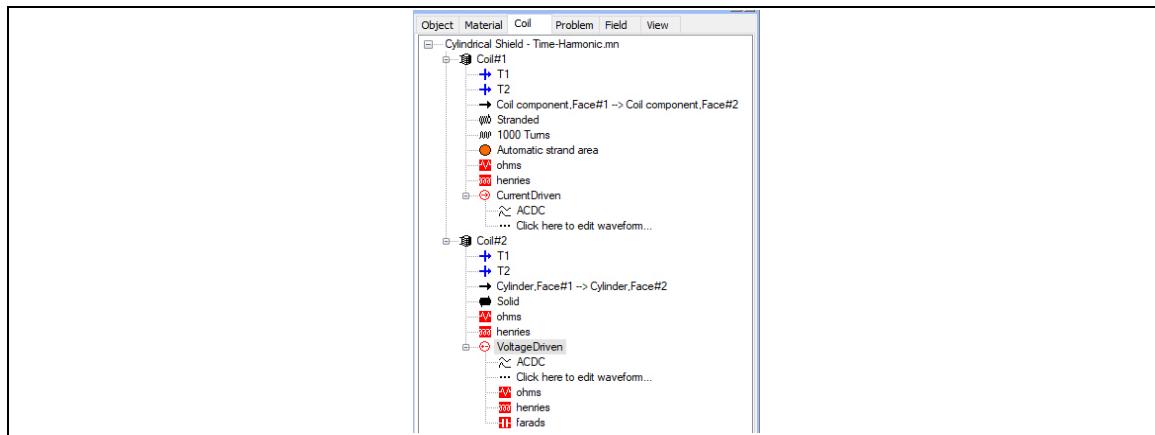
Note The symmetry conditions are such that we would expect no net current in the full cylinder, but some net current in the quarter cylinder. To allow for that, a coil must be made from the cylinder so that its ends can be electrically shorted.

4. On the Model menu, click *Make Simple Coil*.
The coil is listed in the Object page as Coil#2.
5. On the File menu, click *Save*.

11 Edit the coil properties

1. On the *Coil* page of the Project bar, select Coil#1.
2. On the Edit menu, click *Properties*.
The *Coil Properties* dialog opens.
3. Select the Coil Attributes tab (if required) and in the *No. of Turns* box, type **1000**.
4. Select the Waveform tab and type the following values for DC:
Amplitude: **63.7**
5. Click *Apply* to keep the *Coil Properties* dialog open.
6. On the Coil page, select Coil#2.
7. Select the Coil Attributes tab and in the *Type* drop-down list, select **Solid**.
8. In the *Source* drop-down list, select **VoltageDriven** and then click *OK*.

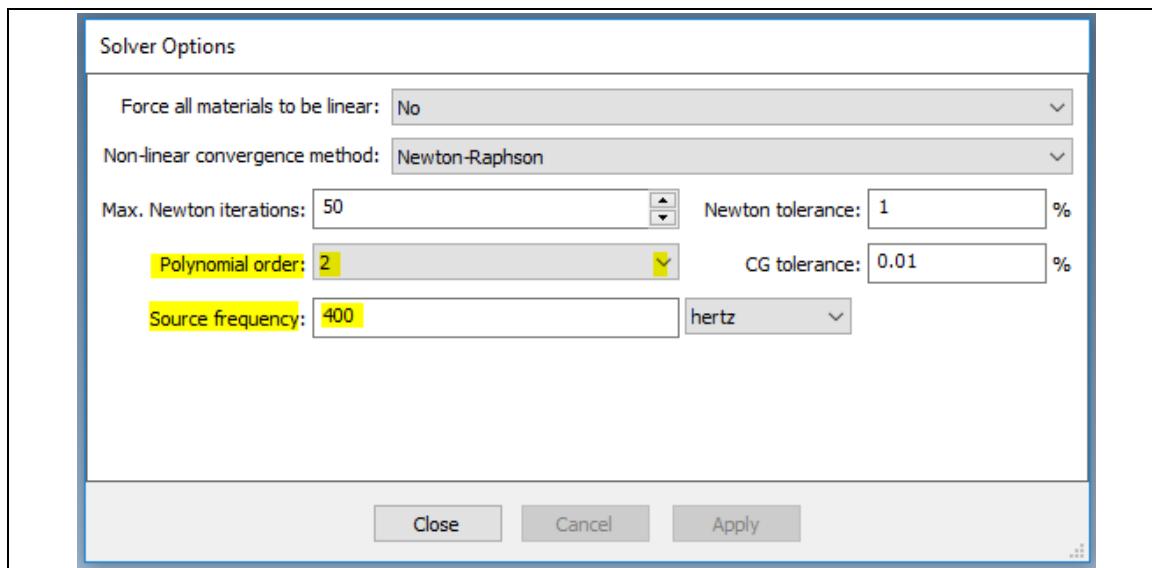
The coil page is automatically updated.



12 Set the source frequency

The source frequency is set in the Solver Options page or the Model Properties page. We will use the former.

1. On the Solve menu, click *Set Solver Options*.
The *Solver Options* dialog appears.
2. Select 2 as the Polynomial order value and in the *Source Frequency* box, type **400**. The default unit is Hertz.



3. Click OK.

13 Change the polynomial order

The potential in each element of the mesh is modeled as a polynomial in the spatial coordinates (x, y). In general, higher orders give greater accuracy, but involve longer solution times. For 2D translational models, the default polynomial order is 1. In this tutorial, the order will be changed to 2.

The polynomial order is set in the Solver Options page or the Model Properties page. We will use the latter.

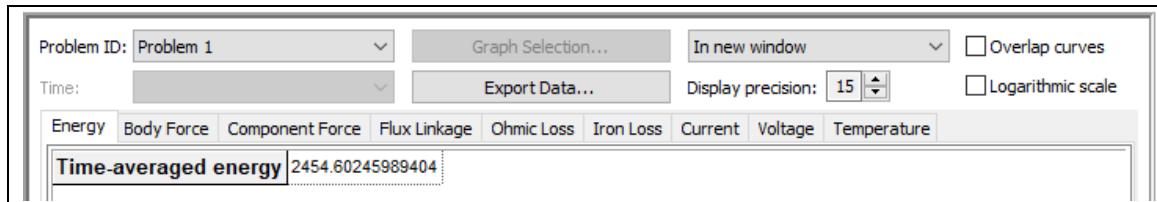
13.1 To change the polynomial order

1. On the Object page, select Cylindrical Shield - Time harmonic.mn.
2. On the Edit menu, click *Properties*.
3. In the Model Properties dialog, select the *Solve* tab.
4. In the *Polynomial order* drop-down list, select **2**.
5. Click OK.
6. On the File menu, click *Save*.

14 Solve

- On the Solve menu, click *Time-Harmonic 2D*.

The cylindrical shield takes about a second to solve (solving time may vary according to computer). The *Time-harmonic 2D Solver Progress* dialog appears briefly and automatically exits when the solution is complete. The Results window then opens.



15 View the solution results

The following results will be reviewed in this section:

- The magnetic flux lines
- The time-averaged Ohmic Loss in each conductor
- The B field magnitude near (0, 0)

15.1 View the magnetic flux lines

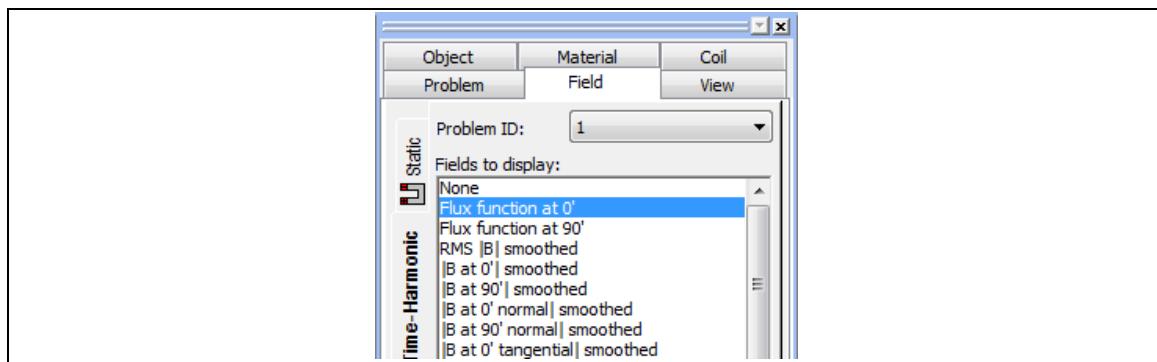
We will display the contours of the magnetic flux function. These contours are the magnetic flux lines (lines that are everywhere parallel to the flux density vector).

- Before viewing the contour plot, switch back to the View window by clicking the View tab

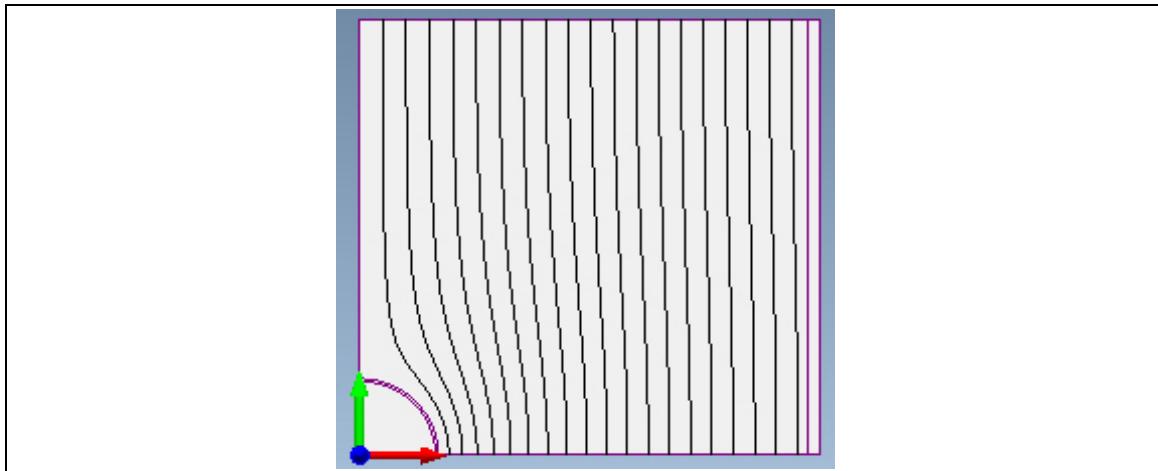


located at the bottom of the window.

- On the Preset View toolbar, click [Show XY (+Z)].
- On the Project bar, select the Field tab.



4. At the bottom of the Field page, select the *Contour* tab.
5. In the *Fields to display* list, select **Flux Function at 0°**.
6. Select the *Shaded* tab.
7. At the top of the *Fields to display* list, select **None**.
8. At the bottom of the Field page, press *Update View*.



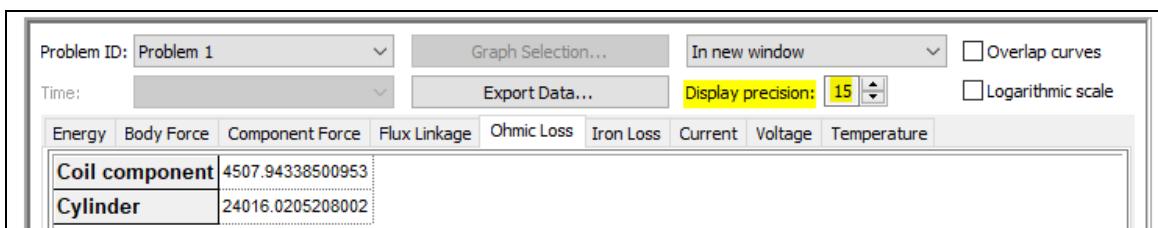
The entire contour plot should be displayed if you have left the Automatic View All tool active, otherwise click the tool again.

15.2 View the time-averaged ohmic losses

Click the Results tab  to switch back to the Results window.

- Select the *Ohmic Loss* tab.

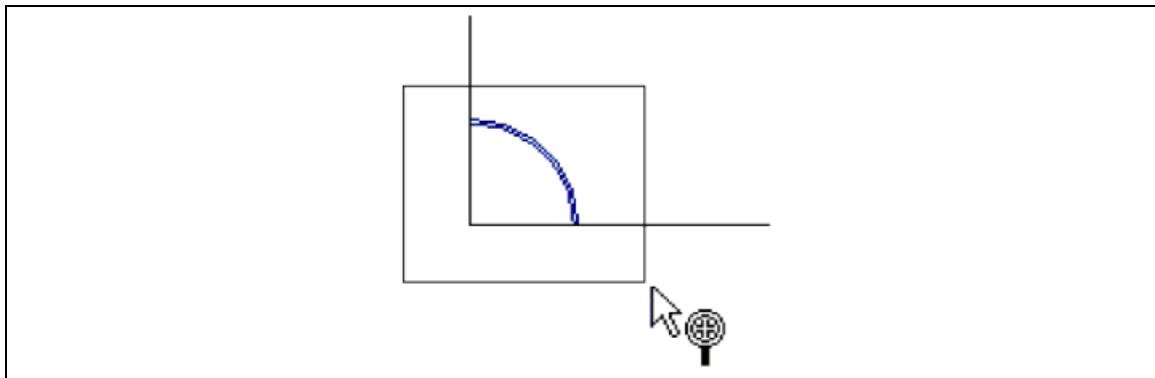
The *Ohmic Loss* page displays the time-averaged Ohmic loss in each conducting component in the model. Note that the Display precision is set to 15.



15.3 Set the color interpolation and style of the shaded plots

This procedure will set the shaded plots to smooth instead of discrete, which is the default.

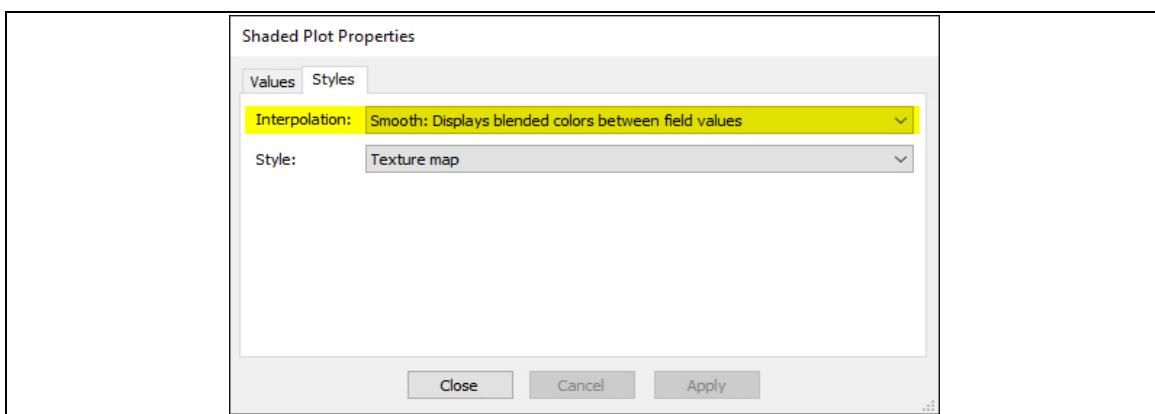
1. On the View menu, click *Default Fields*.
2. With the Examine Model tool still active, hold down the CTRL key and the left mouse button to form a rectangular box around the cylinder.



3. Release the mouse pointer.

The area enclosed by the rectangle is enlarged.

4. On the *View* page of the Project Bar, click *Shaded Plot*.
5. On the Edit menu, click *Properties*.
6. On the Shaded Plot Properties dialog, click the Styles tab.

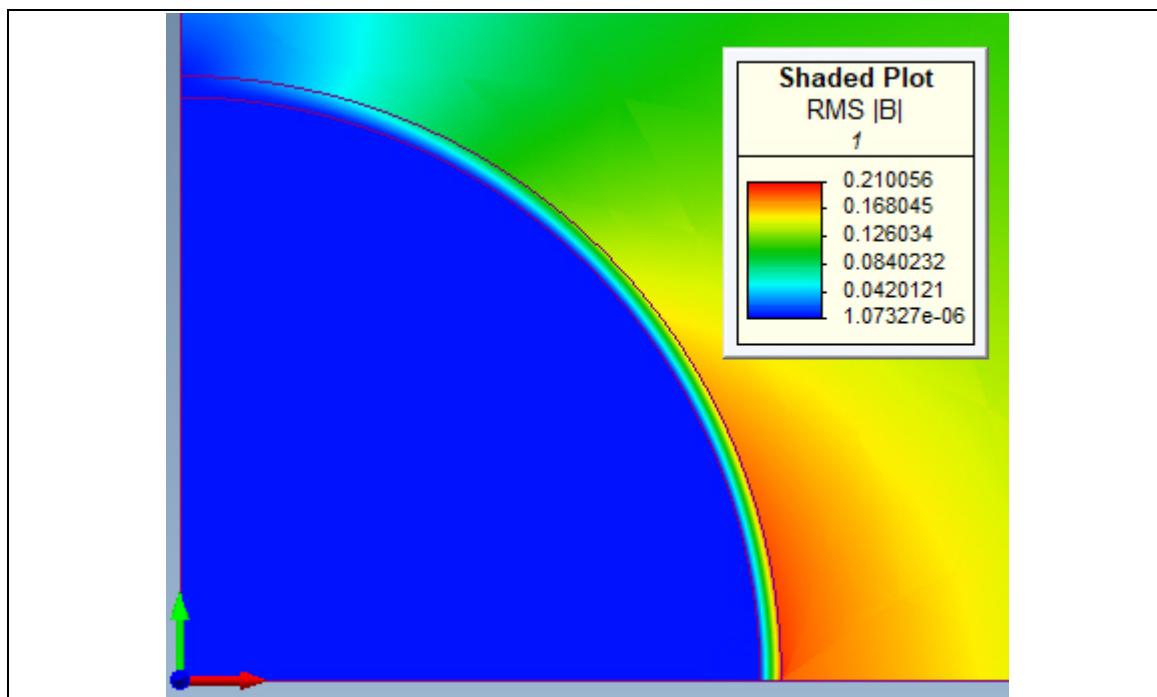


7. From the Interpolation drop-down menu, select "Smooth: Displays blended colors between field values".
8. Click OK.

15.4 View the shaded plot of RMS |B|

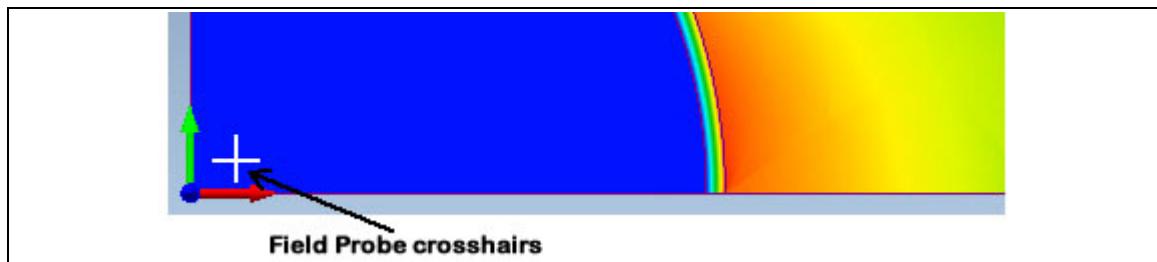
1. On the Project bar, select the *Field* tab.
2. Select the *Contour* tab.
3. In the *Fields to display* list, select **None**.
4. Select the *Shaded* tab.
5. In the *Fields to display* list, select **RMS |B|**.
6. Click **Update View**.

The shaded plot is displayed with a color legend beside it.



15.5 Probe RMS |B| near (0, 0)

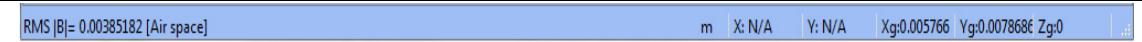
The Field Probe feature allows you to probe for field values, using the mouse to pinpoint a desired location over the solution. The field value and the global coordinates of its location are displayed in the Status Bar. This information can also be written to the Text Output Bar.



15.6 To probe for field values using the mouse

1. On the Tools menu, click *Field Probe* to enable it.
2. Move the mouse (crosshairs) over the solution near (0, 0).

The field value and its specified location on the solution are displayed in the Status Bar.



3. Click the left mouse button over any area of the solution.

The Text Output Bar automatically opens (if it wasn't already opened), and the x, y, and z coordinates of the location on the solution are displayed along with the field value.

X	Y	Z	Field	Value	Component
0.0105372	0.00997404	0	RMS B	0.00385204	Air space
0.00629613	0.00523688	0	RMS B	0.003852	Air space
0.00682626	0.00576323	0	RMS B	0.00385201	Air space
0.00682626	0.00786864	0	RMS B	0.00385189	Air space
0.010007	0.00628958	0	RMS B	0.0038522	Air space
0.010007	0.00628958	0	RMS B	0.0038522	Air space

The example above shows the coordinates, field name, field value, and component for several locations that were clicked upon.

15.7 Save the model

You have now completed the time-harmonic version of the Cylindrical Shield.

1. On the File menu, click *Save*.
2. On the File menu, click *Close*.

16 Summary

In this tutorial, you completed the steps in editing the basic Cylindrical Shield model for a time-harmonic solution. The skills you learned include:

- Drawing with the Keyboard Input bar and creating components
- Creating a new material
- Rotating the display of the model
- Modifying the mesh and defining boundary conditions
- Creating a coil and editing its properties
- Setting the source frequency and polynomial order of the model
- Viewing a Contour plot of the solution
- Viewing the time-averaged ohmic loss in a conductor
- Viewing a Shaded plot and probing it using the Field Probe feature

Tutorial #2

2D Transient

Felix long cylinder

1 Introduction

In this tutorial, the cylindrical shield modeled in *Tutorial #1* is used as the basis for the Felix long cylinder. This model consists of an aluminum hollow cylinder placed in a uniform magnetic field that is perpendicular to the axis of the cylinder and decays exponentially with time.

Properties of the model are edited as follows:

- A circuit consisting of one coil and a current source is created
- An exponential waveform for the current source is defined

After solving, the magnetic flux lines are viewed and animated. The instantaneous ohmic loss of the cylinder is also viewed and graphed.

2 Copy the basic model

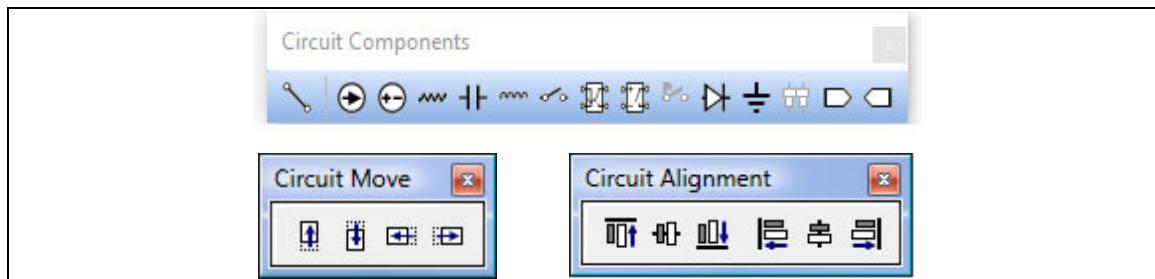
Note If you haven't created the Cylindrical Shield model featured in *Tutorial #1*, please create it by following the *Tutorial #1* procedures, beginning with "Open a new model" and stopping before "Solve", then skip to Step 5 below.

1. On the File menu, click *Open*.
The Open dialog appears.
2. In the *Open* dialog, navigate to the drive and directory that contains the Cylindrical Shield model.
3. Select Cylindrical Shield - Time-Harmonic.mn.
4. Click *Open*.
5. On the File menu, click *Save As*.
6. In the *Save As* dialog box, type **Felix long cylinder** as the name of the model.
7. Choose the drive and directory where you want to place the model.
8. Check the box *Save without meshes and solutions*.
9. Click *Save As*.

3 Create a circuit

The coil in this model is driven by a current source with an exponential waveform.

Although the coil could be driven in the Coil page, it will be driven in the Circuit window to illustrate our circuit capabilities. Circuits are created using the Circuit menu or the three Circuit toolbars. In this tutorial, the Circuit toolbars will be used.

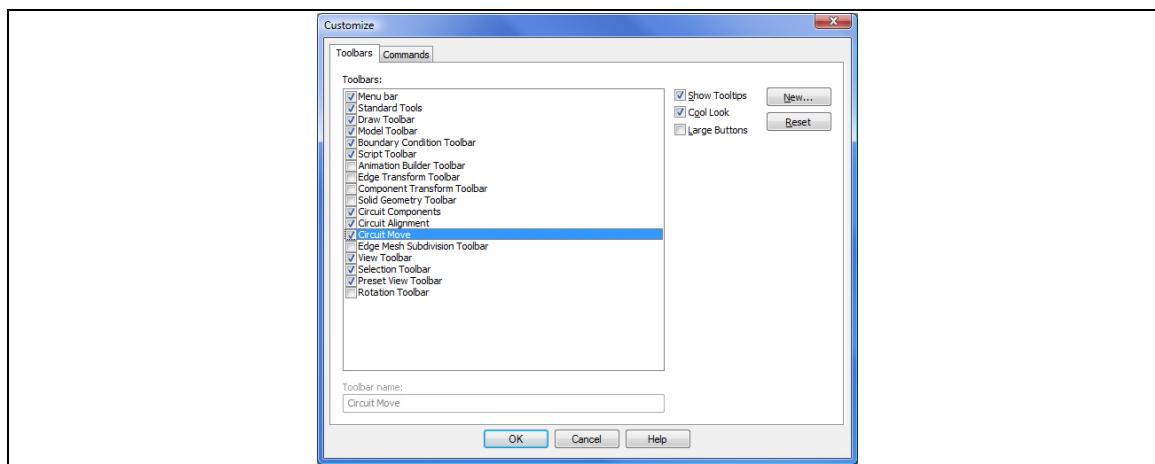


3.1 Display the Circuit toolbars

Note If the circuit toolbars are already displayed, please disregard this procedure.

1. On the Tools menu, click *Customize Toolbars*.

The *Customize* dialog is displayed.



2. From the Toolbars tab, select the following toolbars:

- Circuit Components
- Circuit Alignment
- Circuit Move

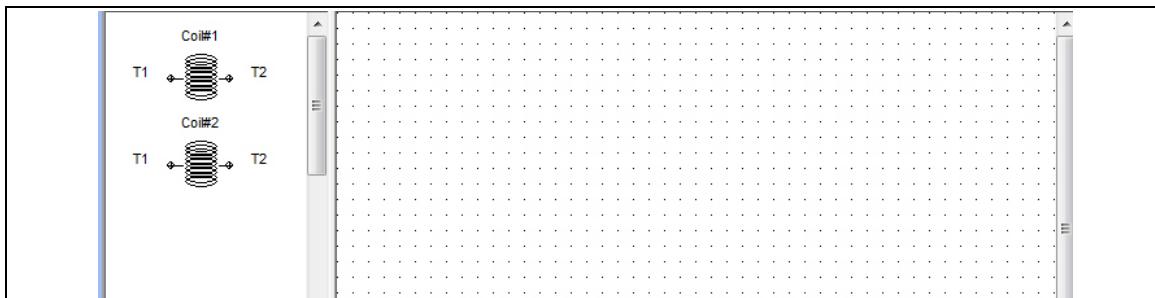
Note The toolbars are displayed as they are selected.

3. Click *OK* to close the Customize dialog and to save your changes.

3.2 Create the circuit

1. On the Window menu, click *New Circuit Window*.

A Circuit window opens. The upper left pane of the Circuit window displays the available coils in the model.



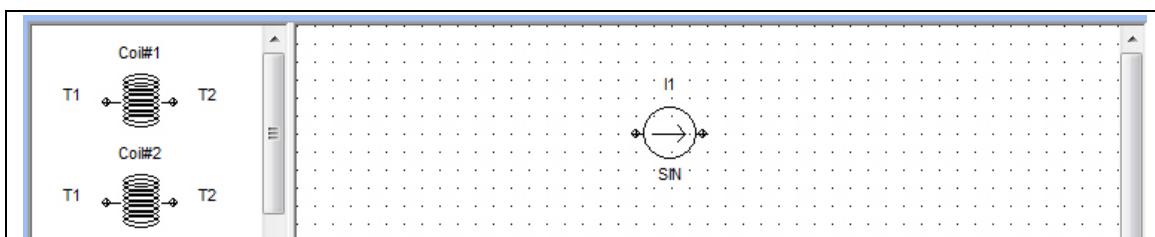
Tip The Circuit tools are also available on the Circuit menu. For example, to add a current source to the circuit, select Current Source on the Circuit menu.

2. On the Circuit Components toolbar, click  (Current Source).

If the Circuit Components toolbar is “grayed out”, click the mouse pointer in the right pane of the Circuit window. The right pane must be activated before the Circuit toolbars are active.

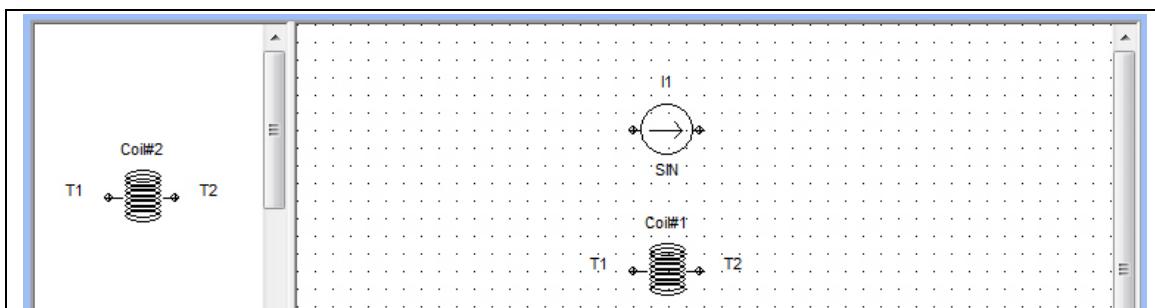
3. Click the mouse pointer in the right pane of the Circuit window.

A current source is added to the window.



4. Select Coil#1 in the upper left pane of the window, and then drag the coil to the right pane.

5. If necessary, re-size the window by dragging an edge of the window.



6. From the Edit menu, click *Select*.

7. Select Coil#1 and the current source with the mouse pointer. Press the SHIFT key on your keyboard while selecting the components.

Tip You can select multiple objects by holding down the SHIFT or CTRL keys while clicking on additional objects. While holding down CTRL, you can click on an object a second time to de-select it.

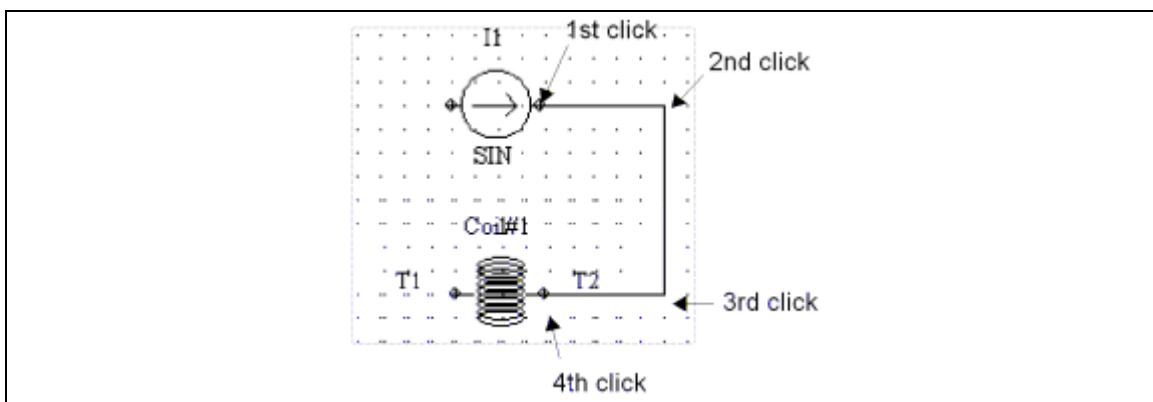
8. On the Circuit Alignment toolbar, click  (Align Center).

The components are aligned at the position of the last selected component.

9. On the Circuit Components toolbar, click  (Connection).

The Connection tool adds connections (wires) between the circuit components.

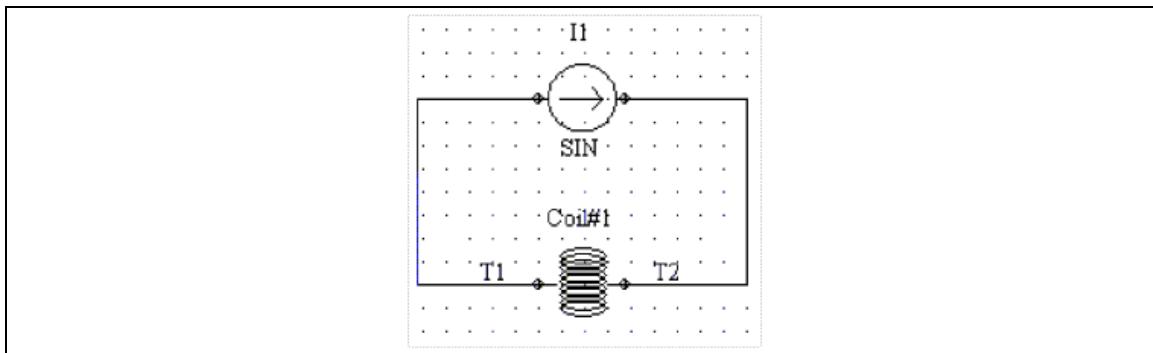
10. Click the mouse pointer on the right terminal of the current source to begin drawing the connection. Continue drawing the connection as shown in the diagram below.



Tip Connections are drawn in the same way as lines are drawn.

11. End the connection on the right terminal of the coil.

12. Draw a second connection as shown in the following diagram.



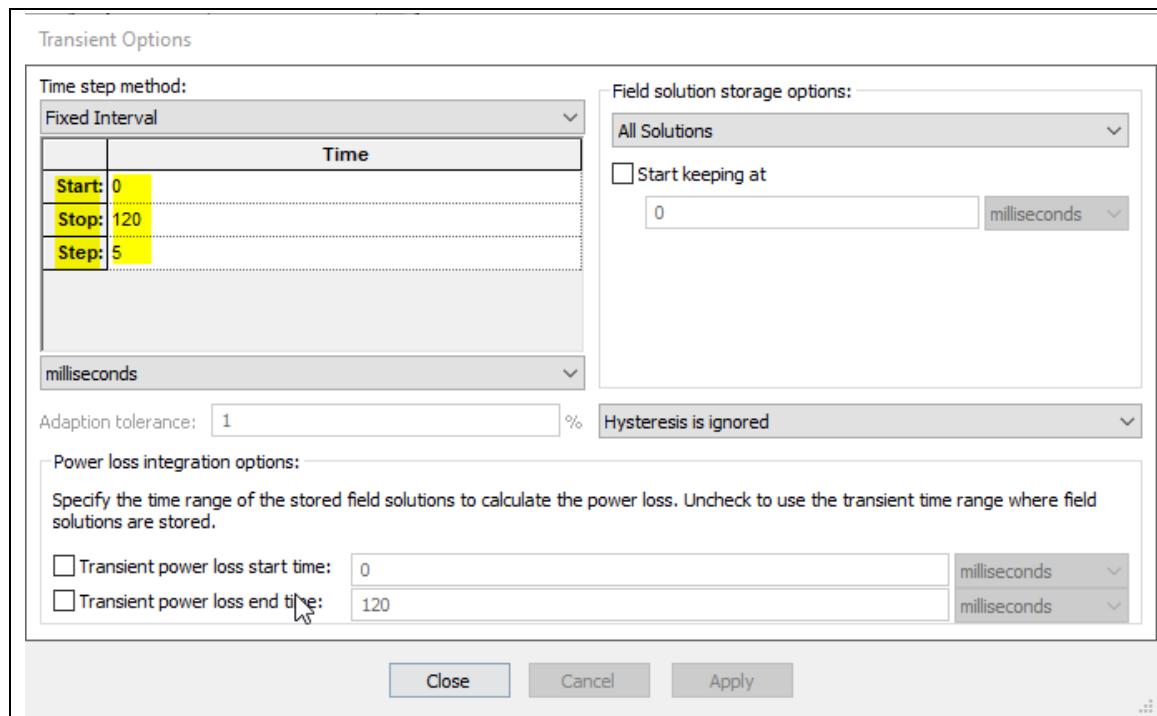
13. On the File menu, click *Save*.

3.3 Set the start, stop and step times

While editing the exponential waveform of the source, the waveform will be previewed between the start and stop times of the simulation. To obtain an accurate preview of the waveform, the simulation time steps should be set before the waveform properties.

Start, stop, and step times can be defined in the Transient Options dialog.

1. On the Solve menu, click *Set Transient Options*.
The *Transient Options* dialog appears.
2. Make sure that *Fixed Interval* is selected as the Time step method, enter in the Time column the values shown below:

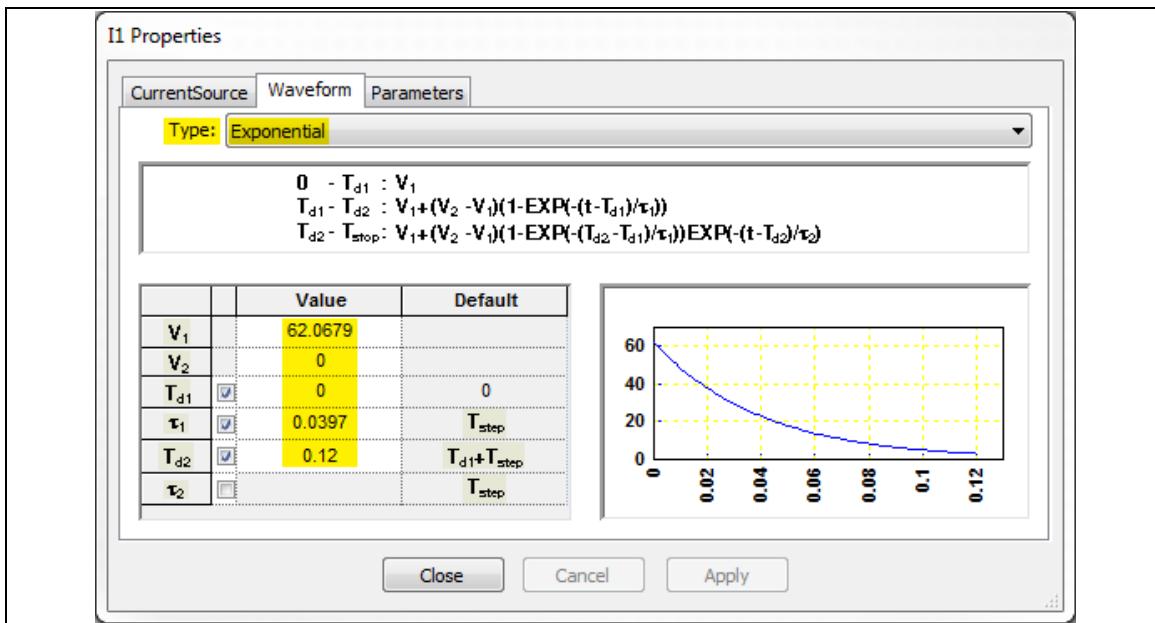


3. Click OK.

3.4 Edit the waveform

1. In the right pane of the Circuit window, right-click the current source.
2. On the pop-up menu, click *Properties*.
The *Current Source (I1) Properties* page appears.
3. Select the Waveform tab.
The *Waveform* property page is displayed.
4. In the Type drop-down list, select **Exponential**.
5. Click the **T_{d2}** checkbox to enable T_{d2} and all preceding optional values.

6. Enter in the Value column the values shown below:



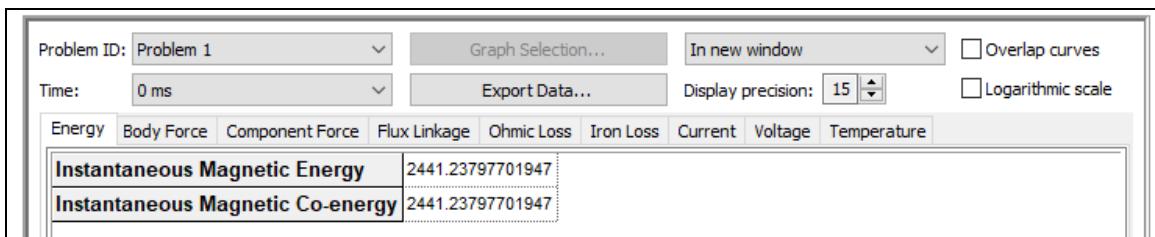
Tip Tooltips in the Value column remind us of the units assumed in the dialog.

7. Click *Apply*.
- The waveform is displayed in the property page.
8. Click *Close*.
9. On the File menu, click *Save*.

4 Solve

- On the Solve menu, click *Transient 2D*.

The *Transient 2D Solver Progress* dialog appears briefly and automatically exits when the solution is complete. The Results window then opens.



5 View the solution results

The following results will be reviewed in this section:

- The magnetic flux lines at the start time
- An animation of the magnetic flux lines across time
- The ohmic loss of each conducting component at the stop time
- A graph of the ohmic loss across time

5.1 View the magnetic flux lines

We will display the contours of the magnetic flux function. These contours are the magnetic flux lines (lines that are everywhere parallel to the flux density vector).

1. Before viewing the contour plot, switch back to the View window by clicking the View tab



located at the bottom of the window.

2. On the Project bar, select the *Field* tab.

The *Field* page opens.

3. Select the *Contour* tab (at the bottom of the Field page).

4. In the *Fields to display* list, select **Flux Function**.

Tip To change the default time unit, use the Model Properties page (in the Object page, right-click the name of the model, then select Properties). In the Units page, select the preferred unit for time. The unit can be Hours, Minutes, Seconds, Milliseconds, or Microseconds.

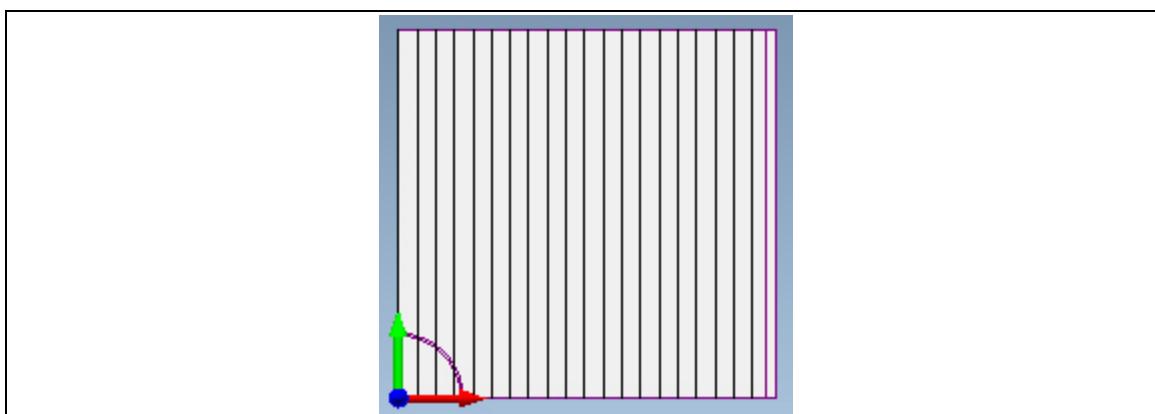
5. In the *Time* drop-down list, make sure **0** is selected (the start time).

6. Select the *Shaded* tab.

7. At the top of the *Fields to display* list, select **None**.

8. At the bottom of the Field page, press *Update View*.

The contour plot is displayed.

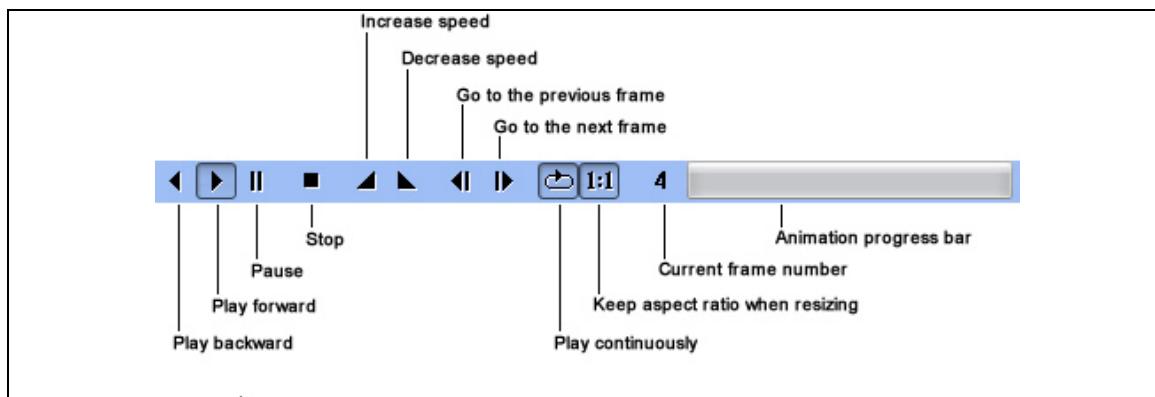


5.2 Animate the magnetic flux lines

An animation is a series of “snapshots”, or frames, that are viewed as a moving image.

1. On the Field page, click  (Animate button).

Please wait a moment while the animation is created. When the animation is complete, an Animation window appears. The animation automatically plays in a loop. Use the Animation Control toolbar to control the playing of the animation.



2. Click the Stop button  when you are finished viewing the animation.

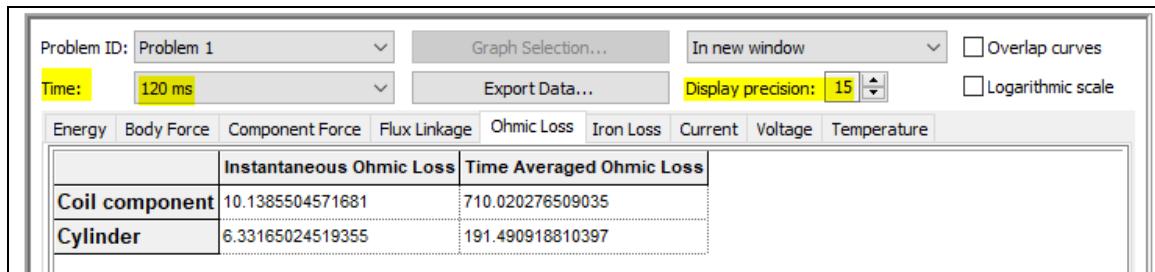
5.3 Save the animation

1. On the File menu, click *Save*.
The *Save As* dialog box appears.
2. In the *File Name* text box, type **Felix long cylinder**.
The animation extension *.ban* is automatically added.
3. Click *Save*.
The animation is saved.
4. On the File menu, click *Close*.
The Animation window closes.

5.4 View the instantaneous ohmic loss

Click the Results tab  to switch back to the Results window.

1. In the *Time* drop-down list, select **120 ms** (the stop time).
2. Select the *Ohmic Loss* tab.

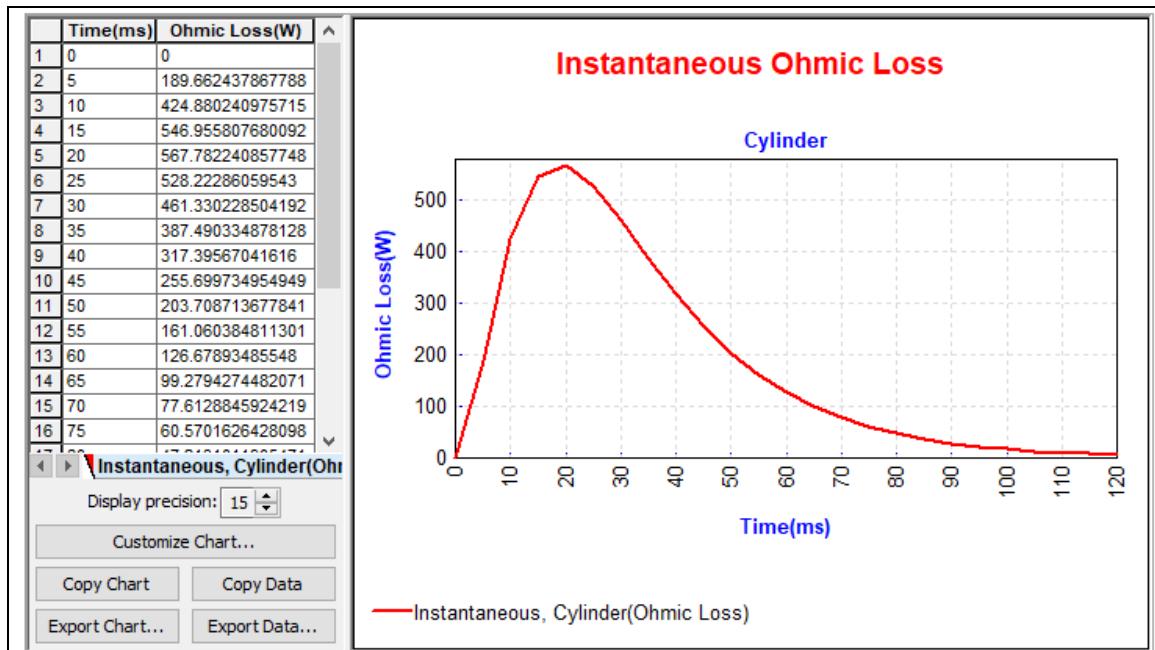


The *Ohmic Loss* page displays the instantaneous Ohmic loss in each conducting component in the model. Note that Display Precision is set to 15.

5.5 Graph the ohmic loss across time

1. Click the mouse pointer in the instantaneous ohmic loss entry (i.e. 6.3316502451935488) for Cylinder.
2. Press the *Graph Selection* button.

A new graph window appears.



5.6 Save the model

You have now completed the Felix long cylinder tutorial.

1. On the File menu, click *Save*.
2. On the File menu, click *Close*.

6 Summary

In this tutorial, you completed the steps in editing the Felix long cylinder for a transient solution. The skills you learned include:

- Creating a circuit
- Defining an exponential waveform
- Viewing a contour plot
- Animating a contour plot
- Viewing the instantaneous ohmic loss of each conducting component
- Graphing the ohmic loss across time.

Tutorial #3

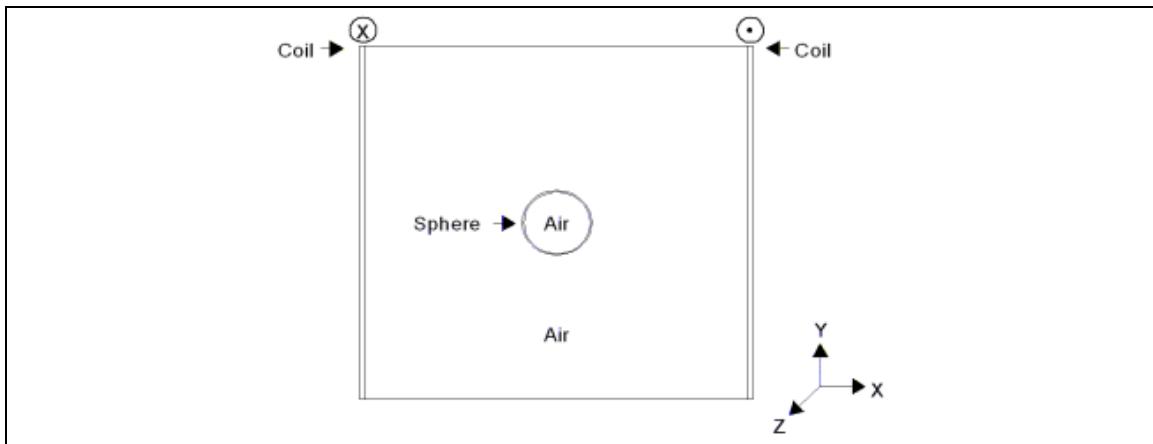
2D

Spherical shield (basic model)

1 Modeling plan

The problem is comprised of a hollow sphere, either ferromagnetic or conducting, lying in a uniform field. The uniform field is provided by an infinitely long cylindrical coil enclosing the sphere.

The problem is the rotational counterpart of the shielding problem considered in *Tutorials #1* and *#2*.

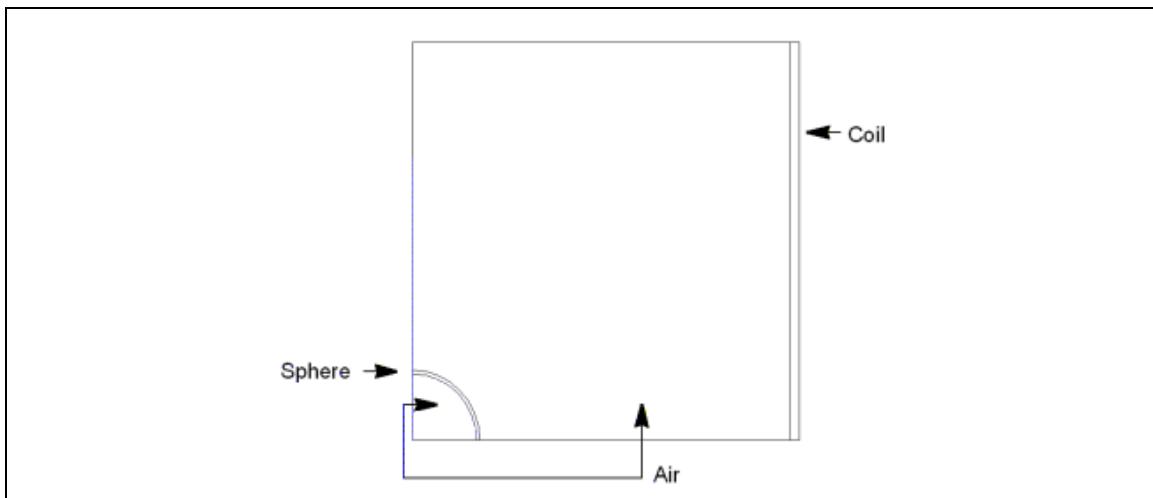


The basic model is built with the sphere assigned the material AIR. In *Tutorial #4*, the model is solved with a ferromagnetic sphere. In *Tutorial #5*, it is solved with a conducting sphere.

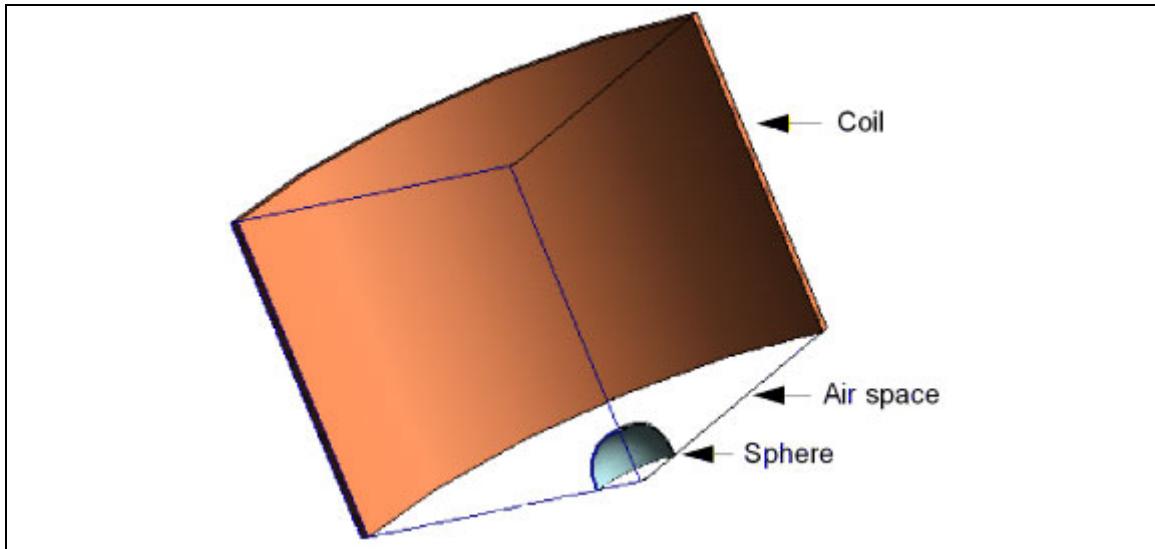
Symmetry conditions are used to model one-half of the problem. The model is built from three components: a sphere, a coil, and an air space.

Note In this tutorial, the default rotational sweep of 90° is used to create the model. However, you could use any sweep angle as only the Z=0 plane is solved for.

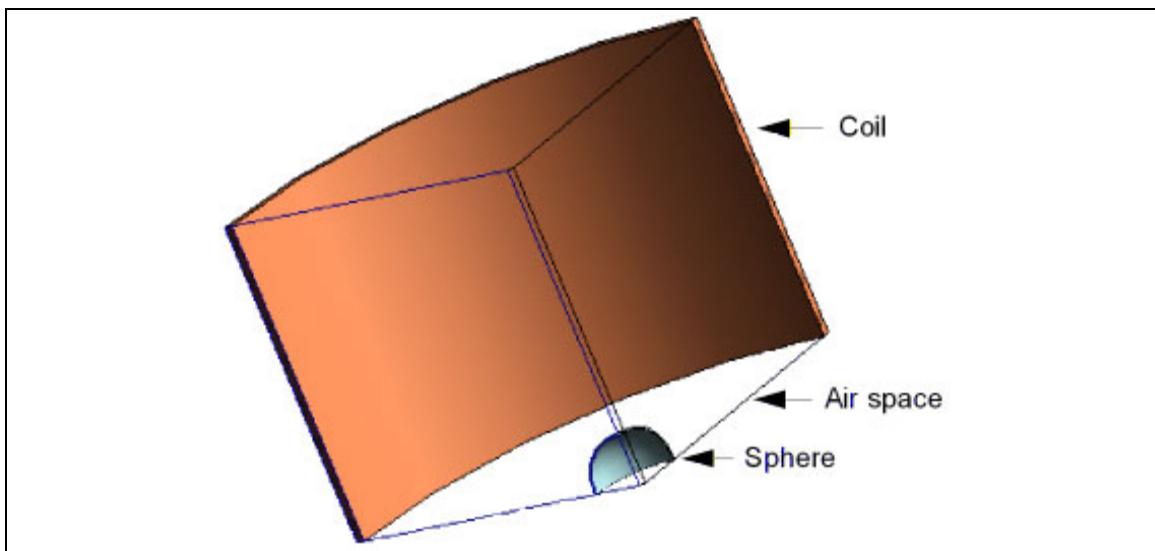
The outline of the model is shown in the following diagram



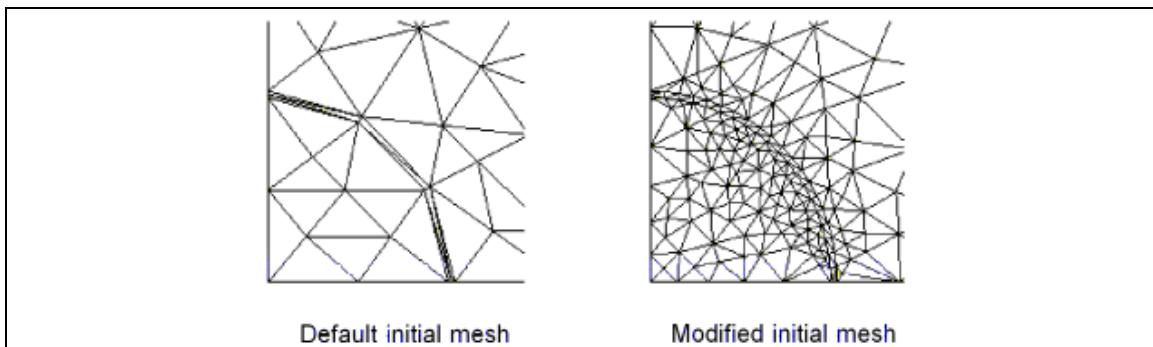
After it is drawn, the outline is swept into components and a coil is created from the coil component.



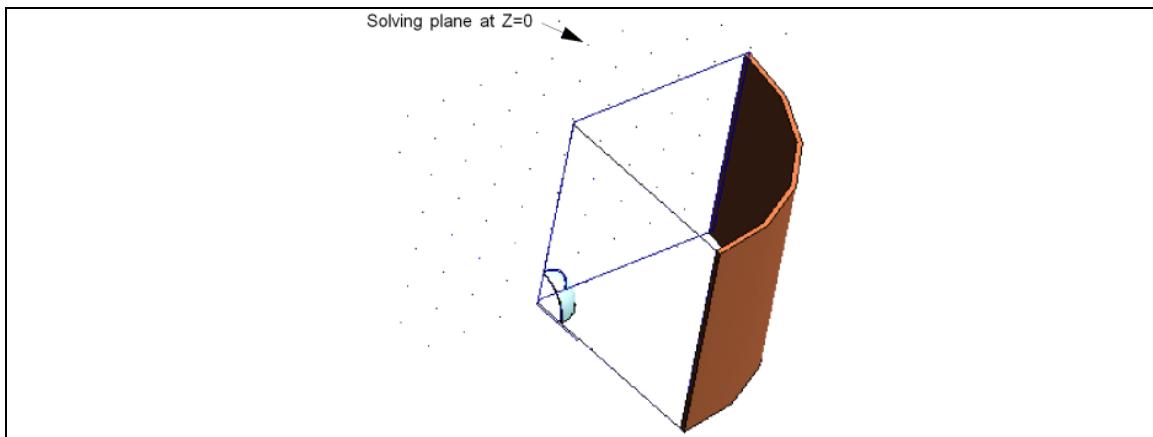
Boundary conditions are applied to indicate symmetry. The Field Normal boundary condition is applied to the top, bottom, and curved boundaries of the model. The default boundary condition, Flux Tangential, is automatically applied to the remaining boundaries.



The density of the mesh will be increased in the area of the sphere to improve solution accuracy.



The model is solved at the XY plane where Z=0 (the default position of the construction slice).



In this tutorial, you will build the basic model with the material of the spherical shell as air. The coil will have the default properties of 1 turn and 0 A.

2 Open a new model

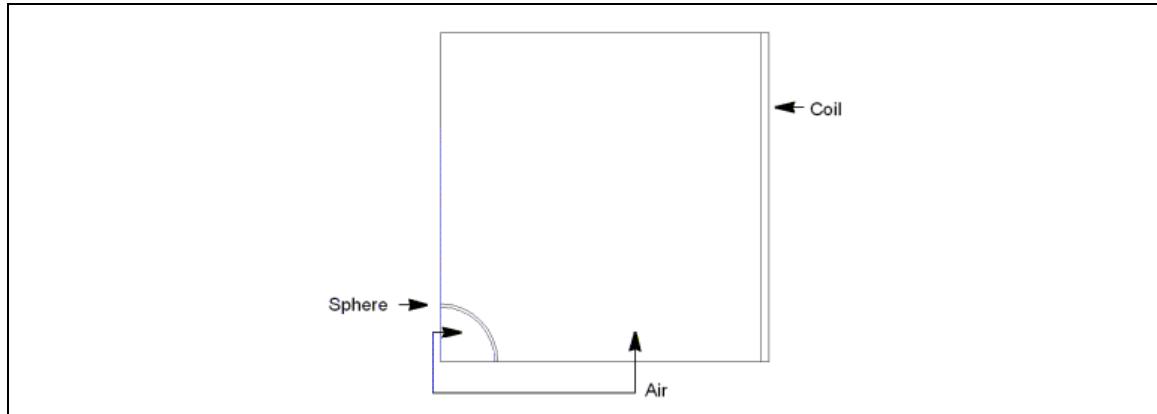
1. Start Simcenter MAGNET.
The Main window appears.
2. If Simcenter MAGNET was already running, select New from the File menu to open a new model.
If you have already used Simcenter MAGNET, the window settings are those that were last active. To maximize the window, click  on its top right corner.

2.1 Name the model

1. On the File menu, click *Save* or *Save As*.
2. In the *Save As* dialog box, type **Spherical Shield** as the name of the model.
3. Choose the drive and directory where you want to place the model.
4. Click *Save As*.

3 Build the geometric model

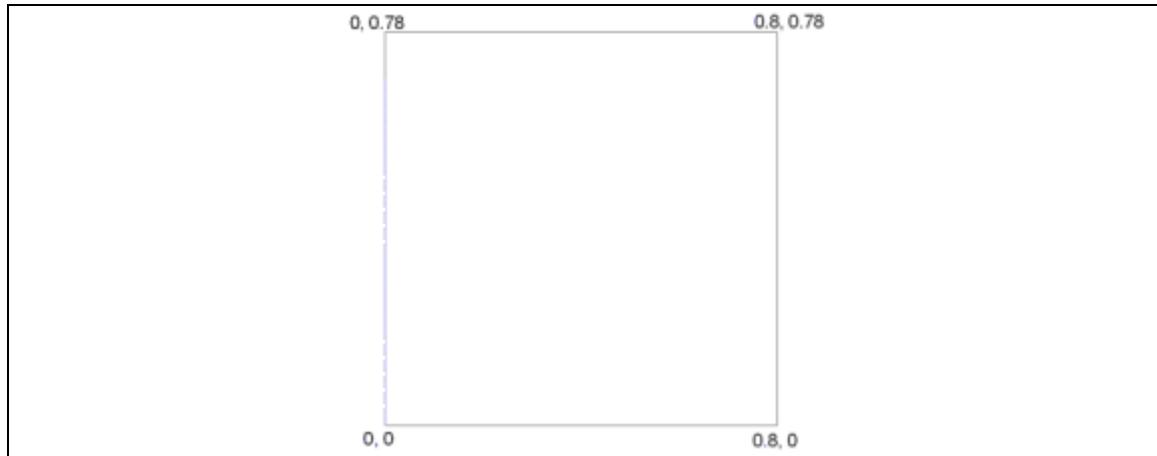
The outline of the model is shown below.



4 Create the air space component

4.1 Draw the outline of the air space

The outline of the air space is shown in the diagram below. Dimensions are in meters, which is the default drawing unit in Simcenter MAGNET.



1. On the Tools menu, select *Keyboard Input Bar* to display it at the bottom of the Main window.



2. See that the Keyboard Input bar is set to (Cartesian) and (Absolute).
3. On the Draw toolbar, click .
4. On the View toolbar, click .

5. In the Keyboard Input bar, enter the following coordinates to draw the air space.

Start coordinates	0, 0	Press ENTER
End coordinates	0.8, 0	Press ENTER
End coordinates	0.8, 0.78	Press ENTER
End coordinates	0, 0.78	Press ENTER
End coordinates	0, 0	Press ENTER

6. Press ESC to stop drawing.

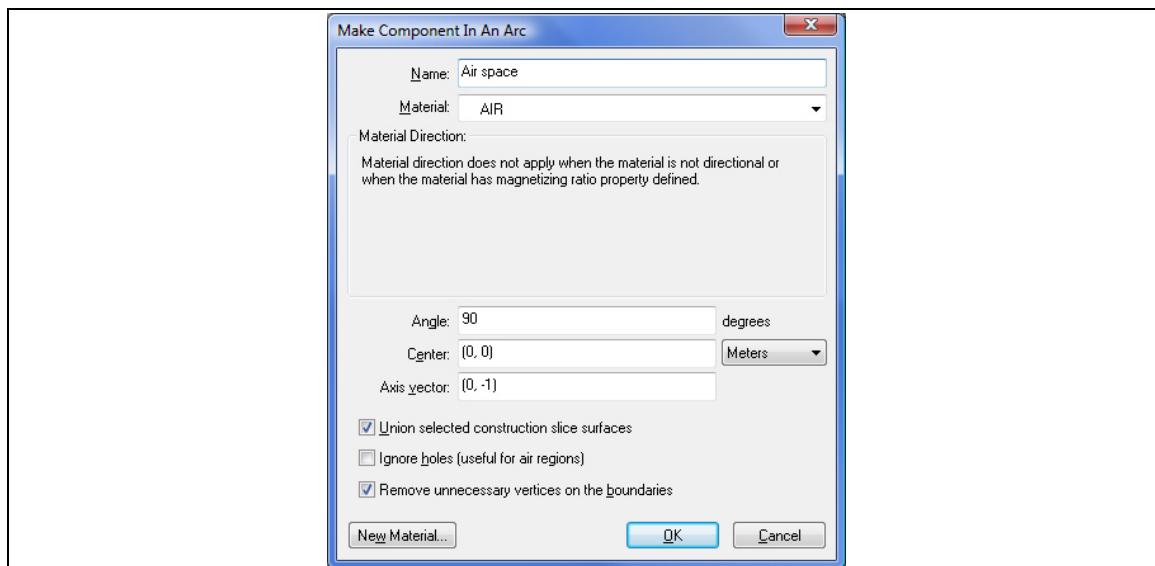
4.2 Make the component of the air space

A component can now be made from the surface that you have drawn. The surface is swept 90° around the negative Y axis (the default rotational sweep).

Note This sweep angle is equivalent to -90° around the positive Y axis.

Components are created using the Make Component dialog box.

1. On the Selection toolbar, click  (Select Construction Slice Surfaces tool). Alternatively, on the Edit menu, click *Select Construction Slice Surfaces*.
2. Click the mouse pointer inside the surface of the air space.
The surface is highlighted when selected.
3. On the Model toolbar, click  (Make Component in an Arc tool).
The *Make Component In An Arc* dialog box appears.



4. Specify the following data in the dialog:

- Name: Air space
- Material: AIR
- Angle: 90 (degrees)
- Center: (0, 0) (Meters)
- Axis vector: (0, -1)

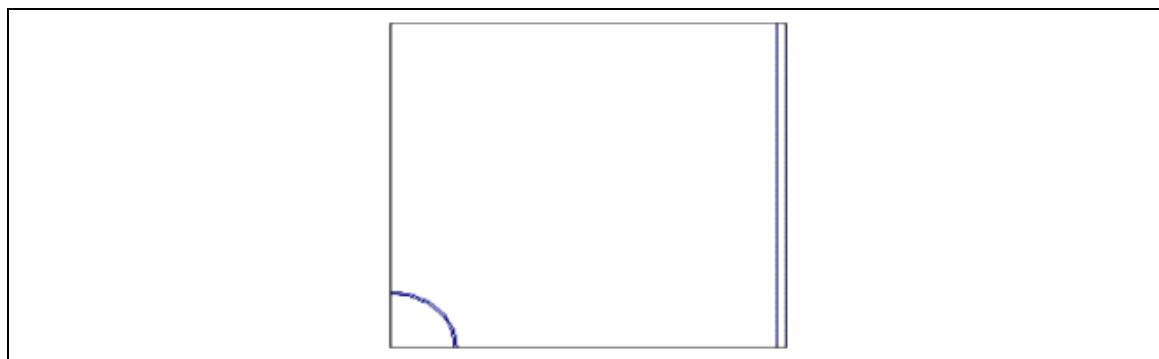
5. Click OK to accept the settings.

The component is created.

6. On the File menu, click *Save*.

5 Create the sphere component

In the outline of the model, the sphere is positioned at the lower left corner. It has a wall thickness of 0.0048 meter and an inner radius of 0.1317 meter.



5.1 Draw the outline of the sphere

1. On the Draw toolbar, click (Center, Start, End).
2. In the Keyboard Input bar, enter the following coordinates for the inner and outer arcs of the sphere, respectively.

Note Arcs are drawn in a counter-clockwise direction.

Center coordinates	0, 0	Press ENTER
--------------------	------	-------------

Start coordinates	0.1317, 0	Press ENTER
-------------------	-----------	-------------

End coordinates	0, 0.1317	Press ENTER
-----------------	-----------	-------------

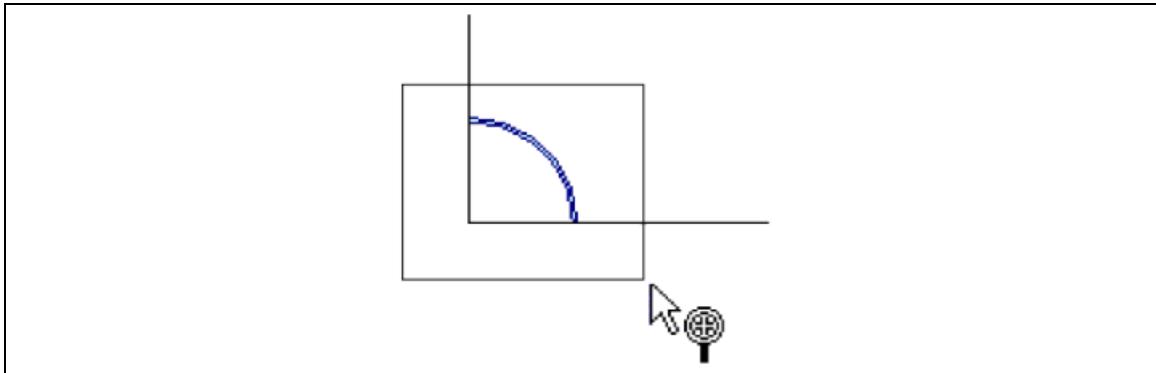
Center coordinates	0, 0	Press ENTER
--------------------	------	-------------

Start coordinates	0.1365, 0	Press ENTER
-------------------	-----------	-------------

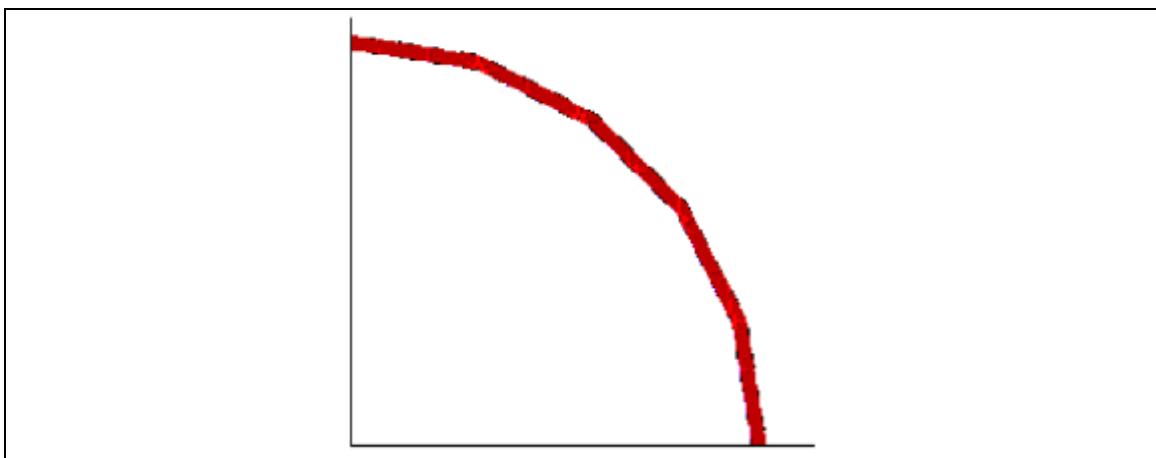
End coordinates	0, 0.1365	Press ENTER
-----------------	-----------	-------------

5.2 Make the component of the sphere

1. On the View toolbar, click  (Examine Model).
2. Hold down the CTRL key and the left mouse button to form a rectangular box around the sphere.



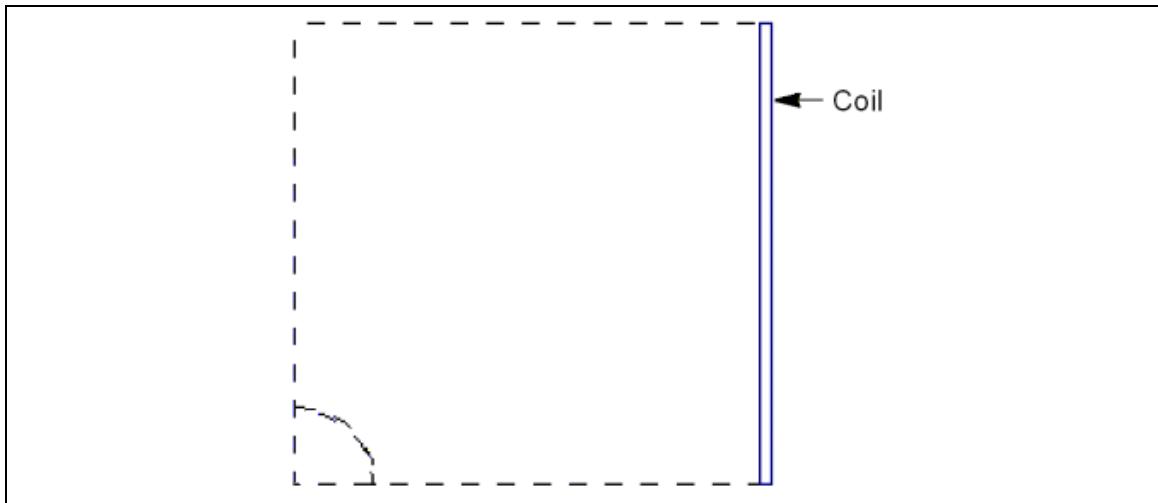
3. Release the mouse pointer.
The area enclosed by the rectangle is enlarged.
4. On the Edit menu, click Select Construction Slice Surfaces.
5. Click the mouse pointer inside the surface of the sphere.
The surface is highlighted when selected.



6. On the Model toolbar, click  (Make Component in an Arc tool).
7. In the Name box, type **Sphere**.
8. In the Material drop-down list, make sure AIR is selected.
This material will be changed in the following chapters, depending on whether the problem setup is for a magnetostatic or time-harmonic solution.
9. Click OK to accept the settings.
The component is created.
10. On the File menu, click Save.

6 Create the coil component

The outline of the coil component is shown below.



6.1 Draw the outline of the coil

1. On the View toolbar, click (View All).
The entire model is displayed.
2. On the Draw toolbar, click (Line drawing tool).
3. In the Keyboard Input bar, enter the following coordinates to complete the drawing of the coil.

Start coordinates	0.78, 0	Press ENTER
End coordinates	0.78, 0.78	Press ENTER
4. Press ESC.

6.2 Make the component of the coil

1. On the Edit menu, click Select Construction Slice Surfaces.
2. Click the mouse pointer inside the surface of the coil.
3. On the Model toolbar, click (Make Component in an Arc tool).
4. In the Name box, enter **Coil component**.
5. In the Material drop-down list, select **Copper: 5.77e7 Siemens/meter**.
6. Click OK to accept the settings and create the component.
7. On the File menu, click Save.

6.3 Delete the lines on the construction slice

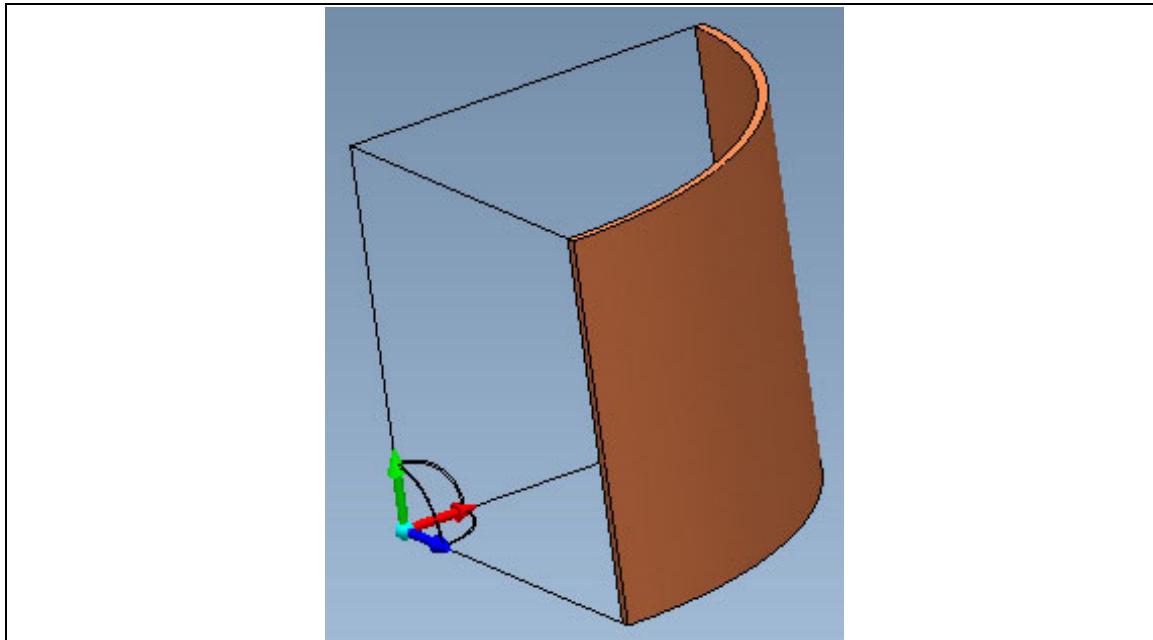
We no longer need the construction slice lines that were used to create the components, so we will proceed to remove them.

1. On the Edit menu, click Select Construction Slice Edges.
2. On the Edit menu, click *Select All*.
3. On the Edit menu, click *Delete*.

6.4 Rotate the display of the model

1. On the View toolbar, click  (Automatic View All).
2. On the View toolbar, click  (Examine Model).
3. Holding down the left mouse button, drag the mouse pointer in one or more of the following directions:
 - Drag up to rotate the display upward.
 - Drag down to rotate the display downward.
 - Drag left to rotate the display toward the left.
 - Drag right to rotate the display toward the right.
4. Release the mouse button.

The display is rotated about the center of the model.



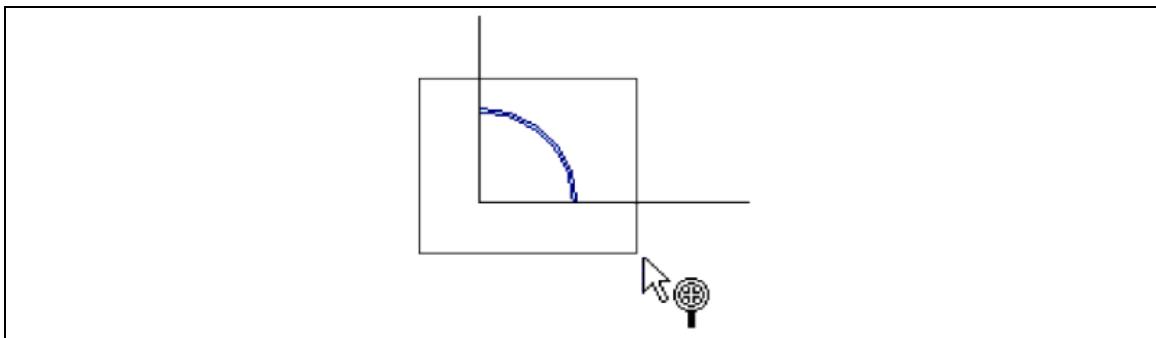
7 Modify the mesh

In the 2D finite element method of analysis, the solution domain is divided into a mesh of triangular elements. The accuracy of the solution depends upon the nature of the field and the size of the mesh elements. In regions where the direction or magnitude of the field is changing rapidly, high accuracy requires smaller elements. One method of increasing the mesh density is to set the *maximum element size* for a component volume or specific faces of a component. The following procedures will demonstrate this method.

7.1 View the initial mesh

Before changing the *maximum element size*, the default initial mesh can be viewed.

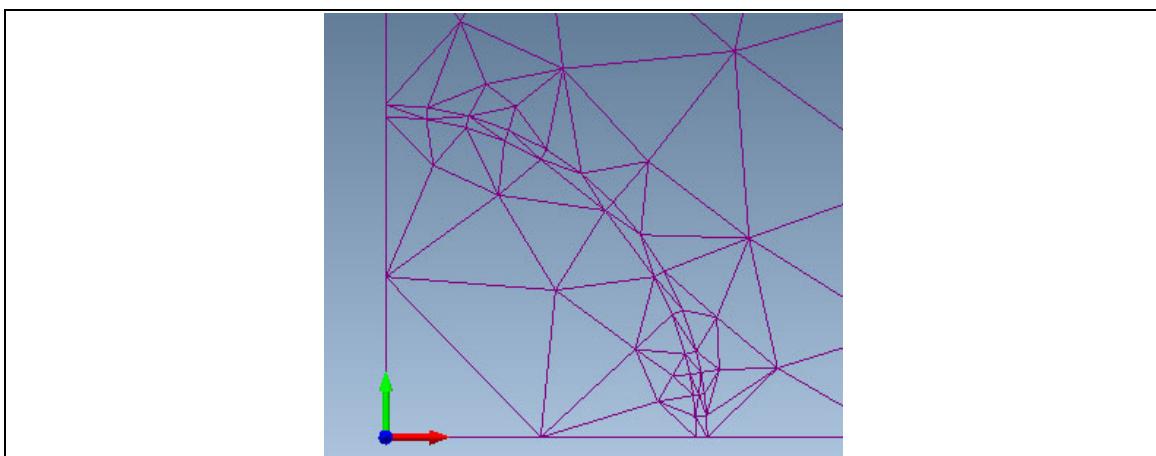
1. On the Preset View toolbar, click  [Show XY (+Z)].
 2. On the View menu, click *Initial 2D Mesh*.
- The initial mesh appears in the View window. It is displayed on the XY plane, at Z=0.
3. With the Examine Model tool still active, hold down the CTRL key and the left mouse button to form a rectangular box around the sphere.



4. Release the mouse pointer.

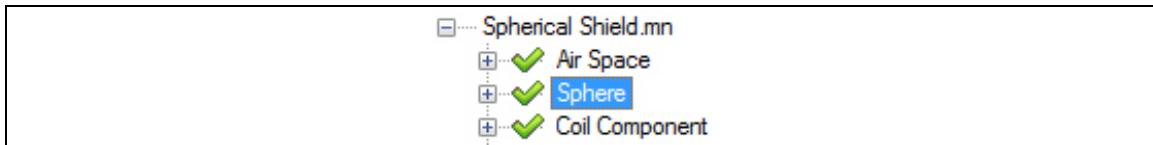
The area enclosed by the rectangle is enlarged.

The mesh should look like the following diagram.

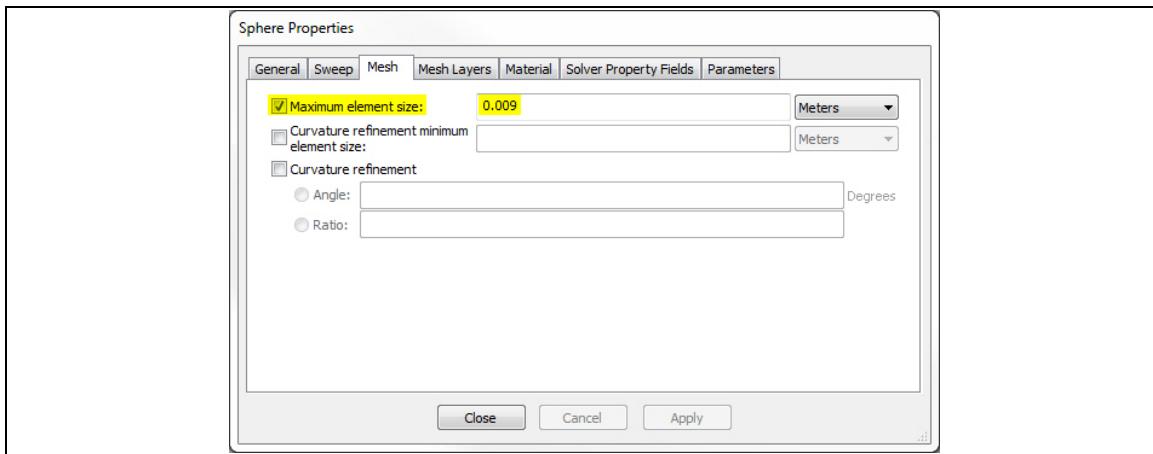


7.2 Set the maximum element size

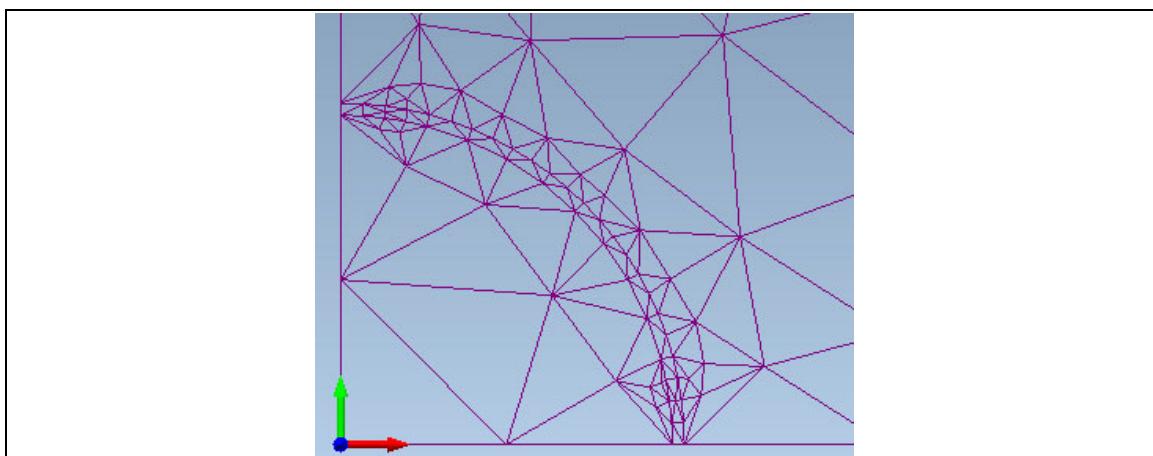
1. In the Object page of the Project bar, select the *Sphere* component.



2. On the Edit menu, click *Properties*.
The Properties dialog appears.
3. Select the *Mesh* tab.
4. Click inside the *Maximum element size* checkbox, and then type **0.009** in the text box.



5. Click OK.
The mesh updates.
6. In the Object page, click the model name to deselect the sphere.
The mesh should look like the following diagram.



7. On the View menu, click *Solid Model*.

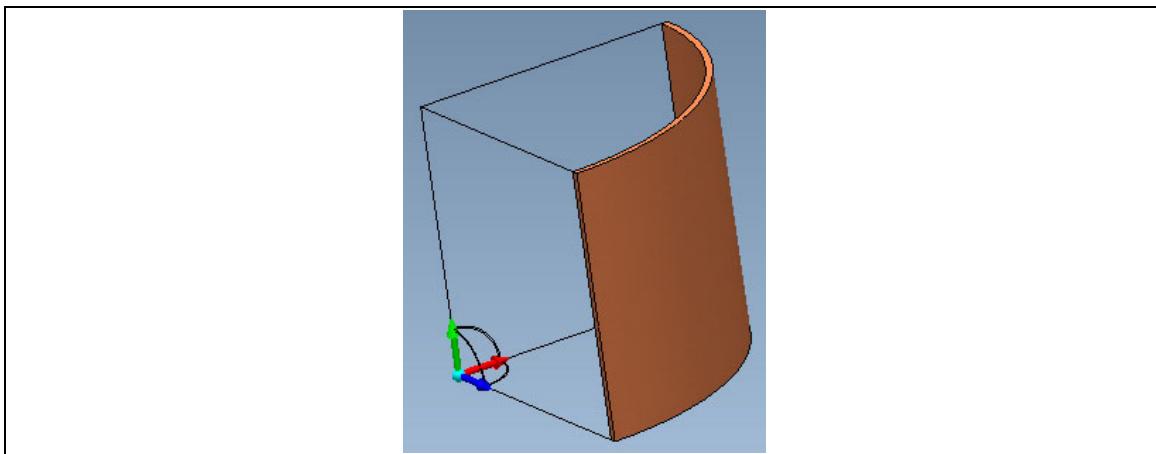
8 Define boundary conditions

The Field Normal boundary condition is applied to three faces of the air space component: the top, bottom, and curved surfaces. The default boundary condition, Flux Tangential, is automatically applied to its remaining surfaces.

The Field Normal boundary condition constrains to zero the tangential component of the magnetic field. The field is made normal (perpendicular) to the boundary. The Flux Tangential boundary condition constrains to zero the normal component of the magnetic flux density. The flux is made to flow tangential to (alongside) the boundary.

8.1 Apply the Field Normal boundary condition

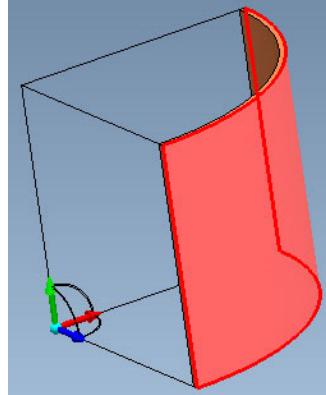
1. On the View toolbar, click  (Automatic View All).
2. With the Examine Model tool still active, hold down the left mouse button and drag the mouse pointer to rotate the model to a 3D view (similar to the diagram below). This rotation will display the surfaces to which the Field Normal boundary condition will be applied.



3. In the Object page, click *Air space, Face#4*.

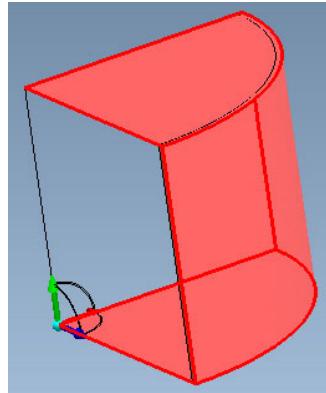


The face is highlighted in the View window.



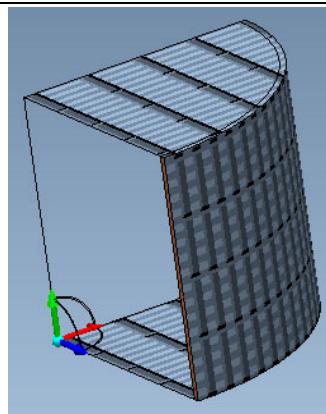
4. While holding down the CTRL key on your computer keyboard, also select *Face#3* and *Face#5* of the *Air space* component.

The faces are highlighted in the View window.



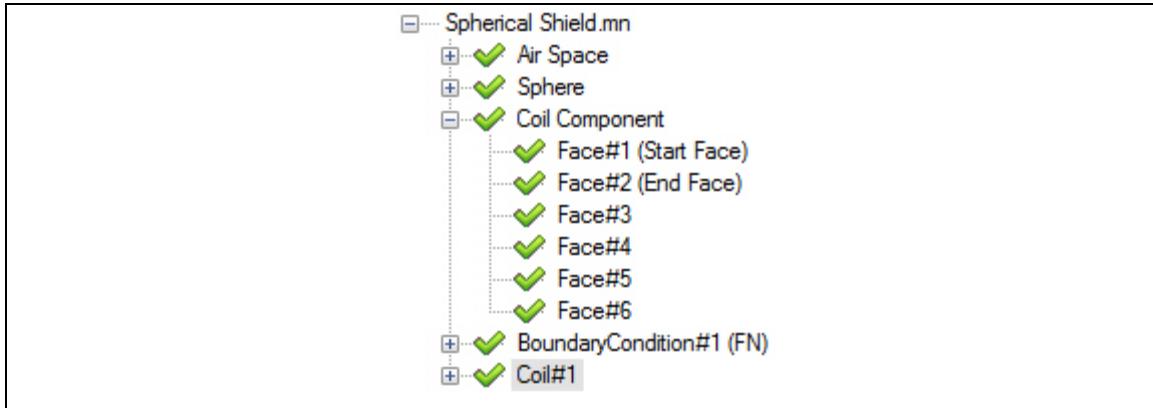
5. On the Boundary Condition toolbar, click (Field Normal).

The *Field Normal* boundary condition is applied to the selected surfaces.



9 Create the coil

1. On the Object page, select *Coil component*.
2. On the Model menu, click *Make Simple Coil*.
The coil is listed in the Object page as *Coil#1*.



3. On the File menu, click *Save*.

10 Summary

You have now completed the base model of the spherical shield. The skills you reviewed in this chapter include:

- Drawing with the Keyboard Input bar
- Creating components
- Rotating the display of the model
- Setting a maximum element size
- Defining boundary conditions
- Creating a coil

To complete the problem definition, continue to one of the tutorials listed below:

- *Tutorials #4: Magnetostatic version: Spherical Shield*
- *Tutorials #5: Time-harmonic version: Spherical Shield*

Tutorial #4

2D Magnetostatic
Spherical shield

1 Introduction

In this tutorial, the spherical shield modeled in *Tutorial #3* is updated with the following properties:

- The material of the sphere is MU3: Relative permeability 1000
- The coil has 1000 turns with a current per turn of 63.7 A

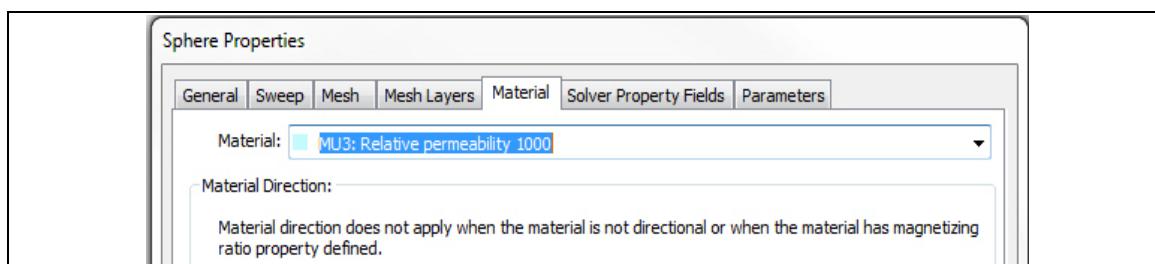
After solving, the magnetic flux lines are viewed, and the B field magnitude in the center of the sphere is obtained.

2 Copy the basic model

1. On the File menu, click *Open*.
The *Open* dialog appears.
2. Navigate to the drive and directory that contains the Spherical Shield model.
3. Select Spherical Shield.mn.
4. Click *Open*.
5. On the File menu, click *Save As*.
6. In the *Save As* dialog box, enter **Spherical Shield - Magnetostatic** as the name of the model.
7. Choose the drive and directory where you want to place the model.
8. Check the box *Save without meshes and solutions*.
9. Click *Save As*.

3 Change the material of the sphere

1. On the Object page, select the *Sphere* component.
2. On the Edit menu, click *Properties*.
The Sphere Properties dialog is displayed.
3. Select the *Material* tab.
The dialog shows that the material currently applied to the component is AIR.
4. In the *Material* drop-down list, select **MU3: Relative permeability 1000**.



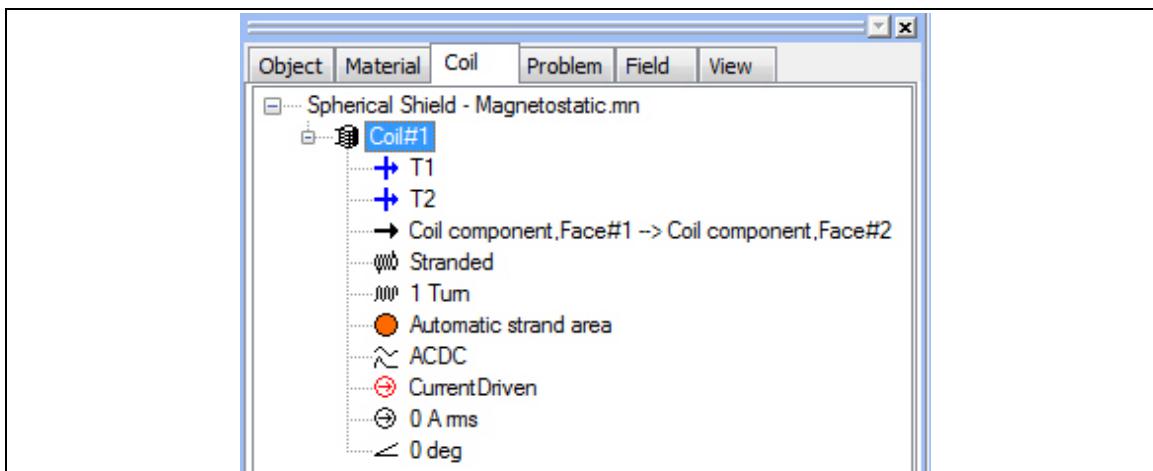
5. Click OK.
The material is applied to the component.

4 Edit the coil properties

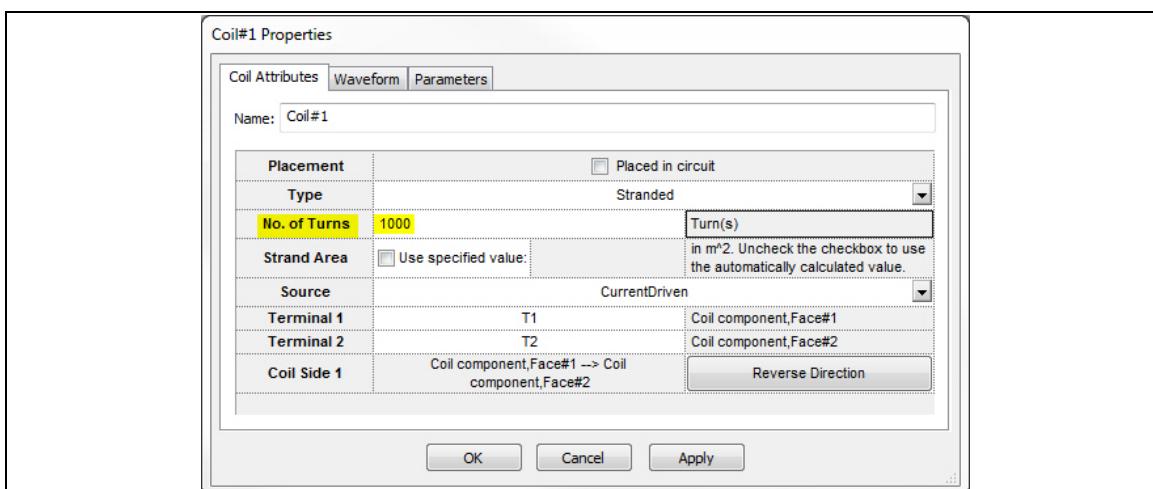
1. On the Project bar, select the *Coil* tab.

The *Coil* page is displayed.

Tip Re-size the Coil page by dragging the right side of the Project bar.



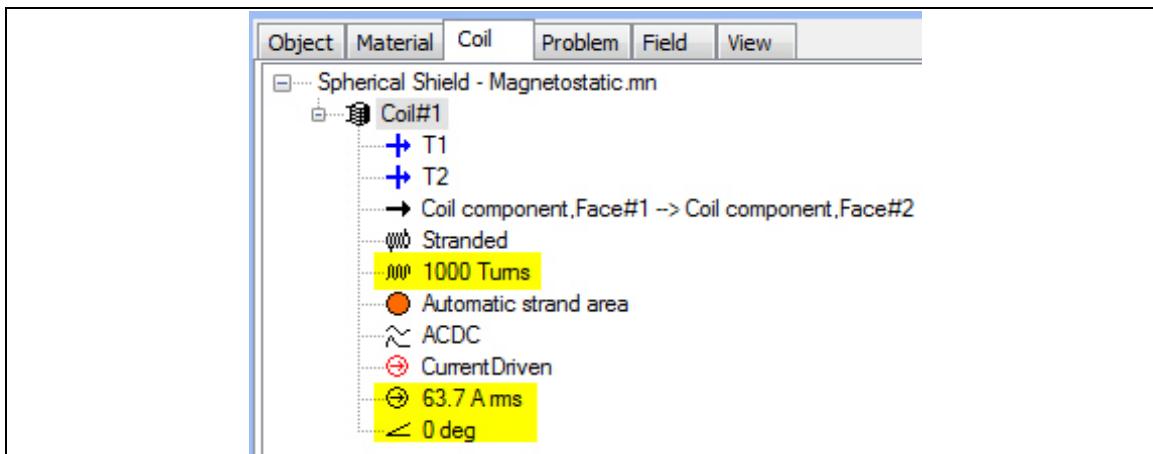
2. Select the name of the coil (Coil#1).
3. On the Edit menu, click *Properties*.
The Coil Properties dialog appears.
4. Select the Coil Attributes tab (if not already selected) and in the *No. of Turns* box, type **1000**.



5. Select the Waveform tab and type the following values for DC:
Amplitude: **63.7**

6. Click OK.

The Coil page is automatically updated.



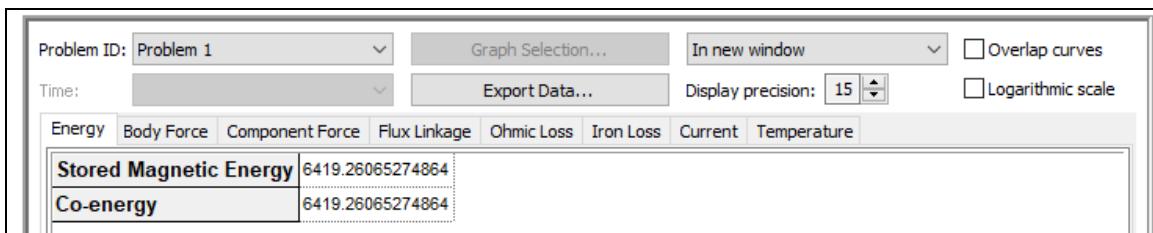
7. Click in the view outside of the model to deselect the coil component.

8. Click *Save*.

5 Solve

- On the Solve menu, click *Static 2D*.

The *Static 2D Solver Progress* dialog appears briefly and automatically exits when the solution is complete. The Results window then opens.



6 View the solution results

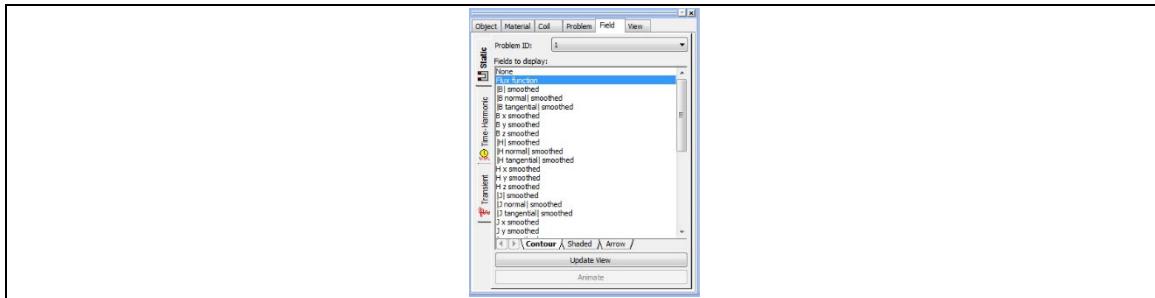
The following results will be reviewed in this section:

- The magnetic flux lines
- The B field magnitude in the sphere

6.1 View the magnetic flux lines

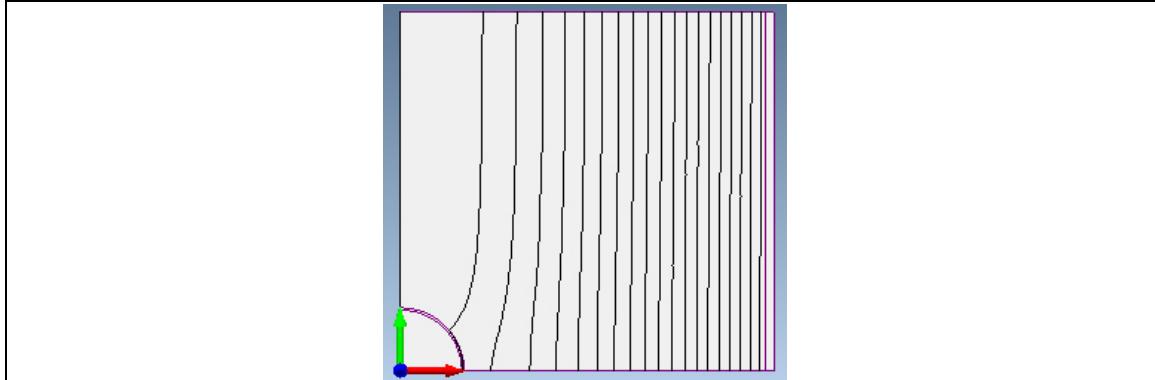
We will display the contours of the magnetic flux function. These contours are the magnetic flux lines (lines that are everywhere parallel to the flux density vector). Close spacing of the lines indicates a rapidly varying flux function which corresponds to a high value of flux density.

1. Before viewing the contour plot, switch back to the View window by clicking the View tab  located at the bottom of the window.
2. On the Project bar, select the *Field* tab.
The *Field* page is displayed.



3. Select the *Contour* tab located near the bottom of the Field page.
4. In the *Fields to display* list, select **Flux Function**.
5. Select the *Shaded* tab.
6. At the top of the *Fields to display* list, select **None**.
7. At the bottom of the Field page, press *Update View*.

The contour plot should look like the following diagram.

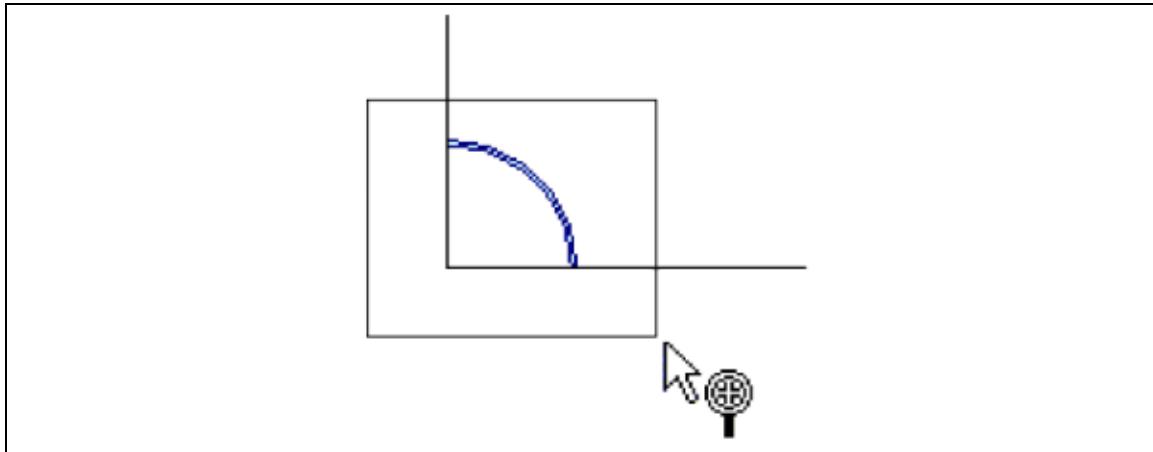


Tip If the View window is not displaying the entire contour plot, click  (View All) on the View toolbar.

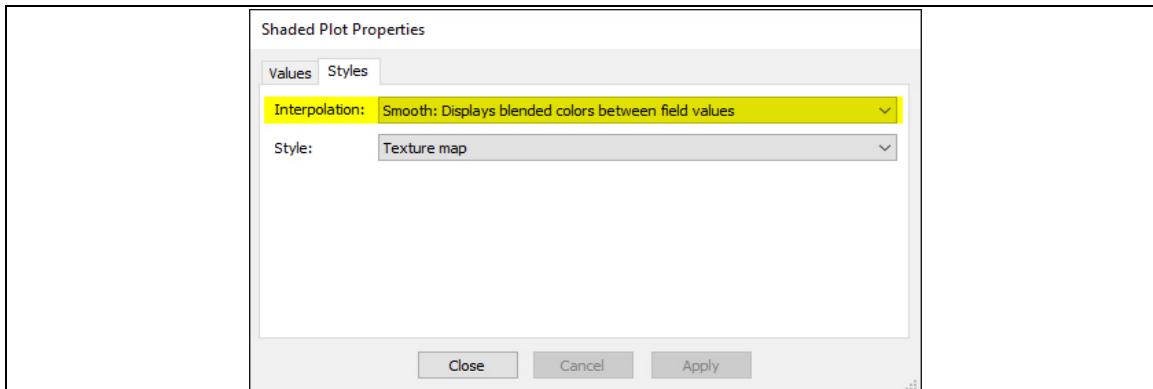
6.2 Set the color interpolation and style of the shaded plot

This procedure will set the shaded plots to smooth instead of discrete, which is the default.

1. On the View menu, click *Default Fields*.
2. With the Examine Model tool still active, hold down the CTRL key and the left mouse button to form a rectangular box around the sphere.

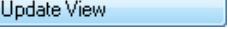


3. Release the mouse pointer.
The area enclosed by the rectangle is enlarged.
4. On the Project Bar, select the *View* tab.
5. From the View tree, click *Shaded Plot*.
6. On the Edit menu, click *Properties*.
The Shaded Plot Properties page appears.

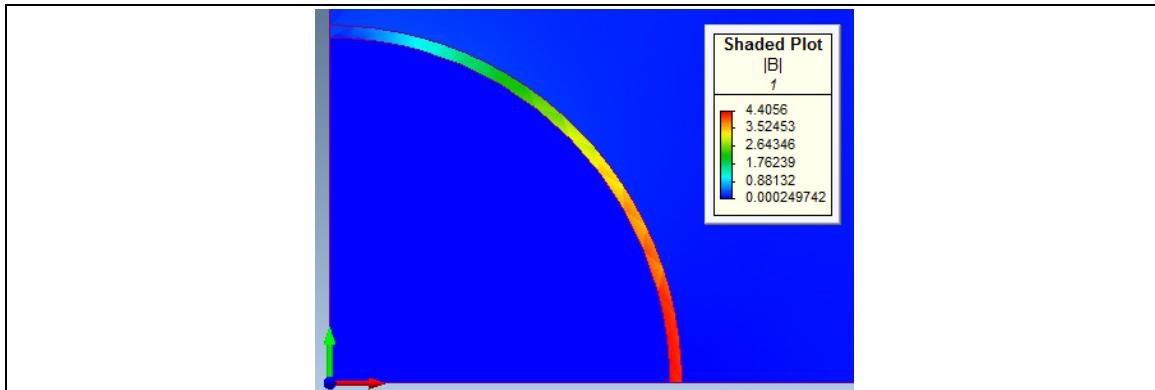


7. Click the Styles tab and then from the Interpolation drop-down menu, select "Smooth: Displays blended colors between field values".
8. Click OK.

6.3 View the shaded plot of $|B|$

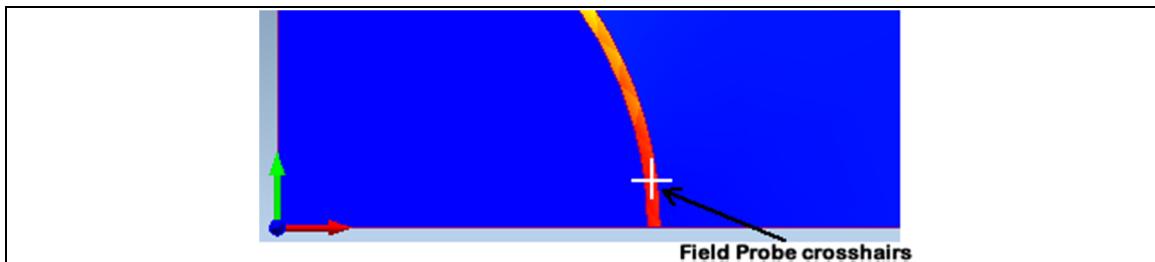
1. On the Project bar, select the *Field* tab.
2. Select the *Contour* tab.
3. In the *Fields to display* list, select **None**.
4. Select the *Shaded* tab.
5. In the *Fields to display* list, select $|B|$.
6. Click .

The shaded plot is displayed with a color legend beside it.



6.4 Probe $|B|$ in the sphere

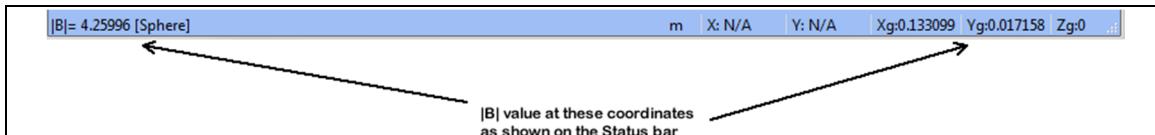
The Field Probe feature allows you to probe for field values, using the mouse to pinpoint a desired location over the solution. The field value and the global coordinates of its location are displayed in the Status Bar. This information can also be written to the Text Output Bar.



6.5 To probe for field values using the mouse

1. On the Tools menu, click *Field Probe* to enable it.
2. Move the mouse (crosshairs) over the solution (as shown in the illustration above).

The field value and its specified location on the solution are displayed in the Status Bar.



- Click the left mouse button over any area of the solution.
- The Text Output Bar automatically opens (if it wasn't already opened), and the x, y, and z coordinates of the location on the solution are displayed along with the field value.

x	y	z	Field	Value	Component
0.0246148	0.132007	0	B	0.775997	Sphere
0.00873352	0.135035	0	B	0.290004	Sphere
0.13334	0.0078305	0	B	4.31392	Sphere
0.127232	0.0453862	0	B	3.7412	Sphere
0.11074	0.078096	0	B	3.4046	Sphere
0.0826425	0.107171	0	B	2.58014	Sphere

The example above shows the coordinates and |B| field values for several locations that were clicked upon.

6.6 Save the model

You have now completed the magnetostatic version of the Spherical Shield.

- On the File menu, click *Save*.
- On the File menu, click *Close*.

7 Summary

In this tutorial, you completed the steps in editing the basic Spherical Shield model for a magnetostatic solution. The skills you learned include:

- Changing the material of a component
- Editing the properties of a coil
- Solving with the default options
- Viewing a Contour plot of the solution
- Viewing a Shaded plot of the solution
- Probing a Shaded plot using the Field Probe feature

7.1 Further exploration

In this tutorial, solution accuracy was helped by an increased mesh density around the sphere. Other possible options include:

- Using Simcenter MAGNET's adaption feature to automatically improve the mesh
- Raising the polynomial order of the entire model to 3 (or higher)

Please see the Help for more information on these features.

Tutorial #5

*2D Time-harmonic
Spherical shield*

1 Introduction

In this tutorial, the spherical shield modeled in *Tutorial #4* is updated with the following properties:

- The material of the sphere is Aluminum 6061 with a conductivity of 2.538e7 S/m
- The source frequency is 400 Hz

After solving, the magnetic flux lines and the ohmic loss are viewed, and the B field magnitude is probed at the center of the sphere.

2 Copy the basic model

1. On the File menu, click *Open*.
The *Open* dialog appears.
2. Navigate to the drive and directory that contains the Spherical Shield - Magnetostatic model.
3. Select Spherical Shield – Magnetostatic.mn.
4. Click *Open*.
5. On the File menu, click *Save As*.
6. In the *Save As* dialog box, enter **Spherical Shield - Time harmonic** as the name of the model.
7. Choose the drive and directory where you want to place the model.
8. Check the box Save without meshes and solutions.
9. Click *Save As*.

3 Create a new material

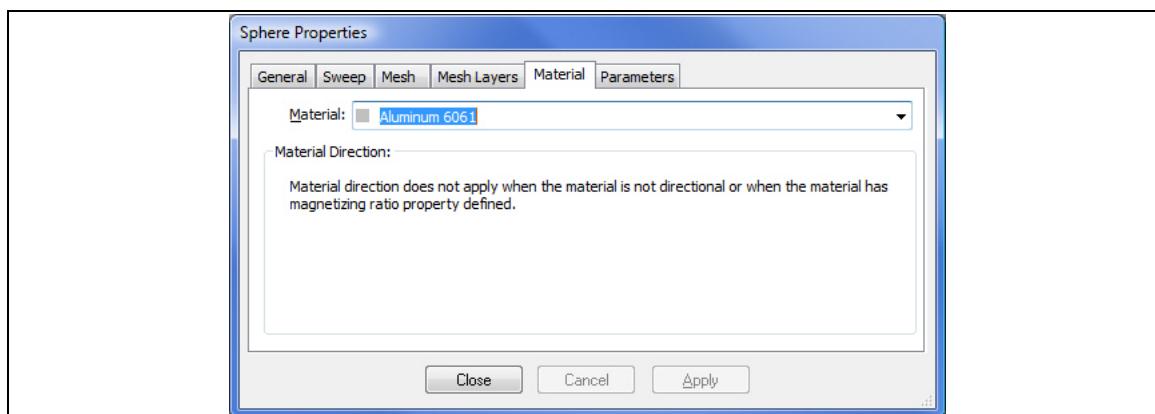
Note If you have already created Aluminum 6061 in Tutorial #1 (Cylindrical Shield - Time-Harmonic tutorial), then you can skip this step and proceed to “Change the material of the sphere”.

1. On the Tools menu, click *New User Material*.
2. On the General page, type the following data:
 - Name: Aluminum 6061
 - Display color: *Click Display Color and select an appropriate color*
3. Click *Next*.
4. On the Options page, select the following:
 - Magnetic Permeability
 - Electric Conductivity

5. Click the *Next* button to advance to the appropriate pages, enter the following values:
 - Temperature (Celsius) = 20
 - Relative Permeability = 1
 - Coercivity (Amps/m) = 0
 - Conductivity (Siemens/m) = 2.538e7
6. Once you have entered all the values, advance to the Confirmation page and click *Finish* to create the new material.

4 Change the material of the sphere

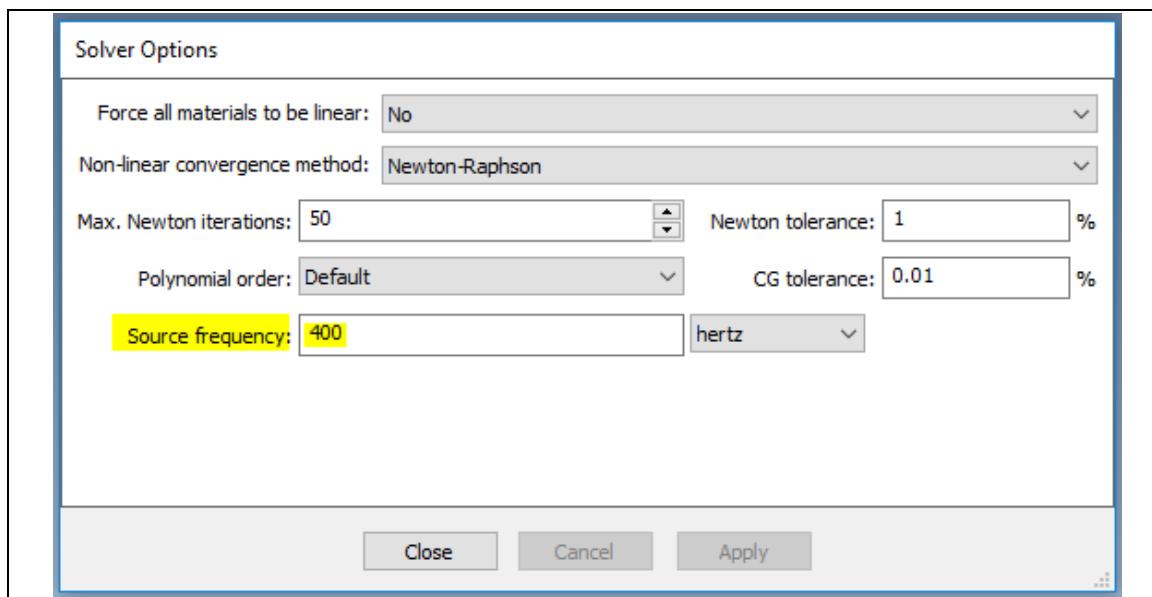
1. On the Object page, select the *Sphere* component.
2. On the Edit menu, click *Properties*.
The Sphere Properties dialog is displayed.
3. Select the *Material* tab.
4. In the *Material* drop-down list, select **Aluminum 6061**.



5. Click OK.
The material is applied to the component.
6. In the Object page, click the model name to deselect the sphere.

5 Set the source frequency

1. On the Solve menu, click *Set Solver Options*.
The *Set Solver Options* dialog appears.
2. In the *Source Frequency* box, type **400**.
The default unit is Hertz.

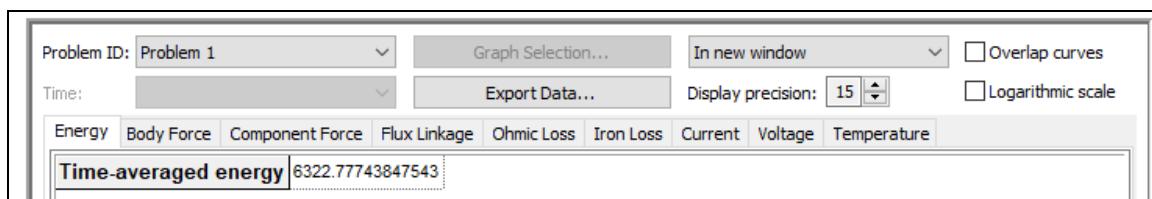


3. Click OK.

6 Solve

- On the Solve menu, click *Time-harmonic 2D*.

The *Time-harmonic 2D Solver Progress* dialog appears briefly and automatically exits when the solution is complete. The Results window then opens.



7 View the solution results

The following results will be reviewed in this section:

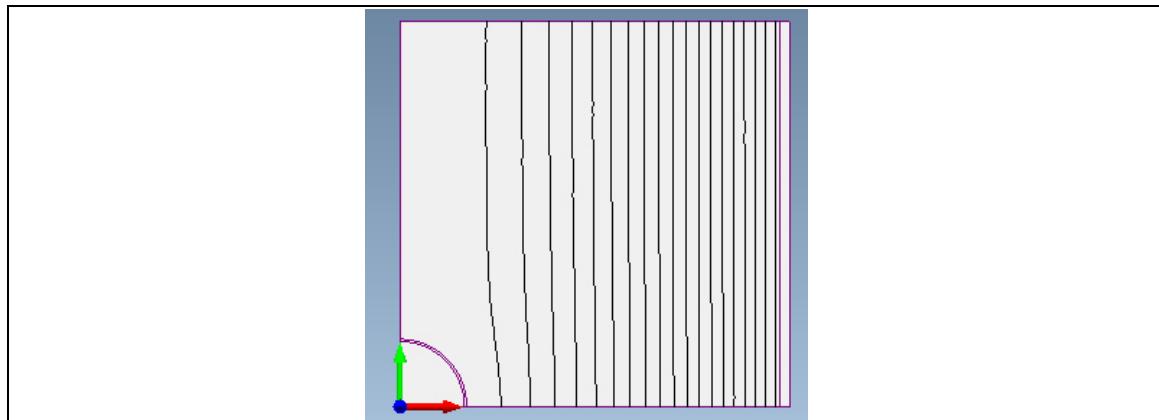
- The magnetic flux lines
- The time-averaged Ohmic loss dissipated in each conductor
- The B field magnitude in the sphere

7.1 View the magnetic flux lines

We will display the contours of the magnetic flux function. These contours are the magnetic flux lines (lines that are everywhere parallel to the flux density vector).

1. Before viewing the contour plot, switch back to the View window by clicking the View tab  located at the bottom of the window.
2. On the Project bar, select the *Field* tab.
The *Field* page is displayed.
3. Select the *Contour* tab (at the bottom of the Field page).
4. In the *Fields to display* list, select **Flux Function at 0°**.
5. Select the *Shaded* tab.
6. At the top of the *Fields to display* list, select **None**.
7. At the bottom of the Field page, press *Update View*.

The contour plot is displayed.



Tip If the View window is not displaying the entire contour plot, click  (View All) on the View toolbar.

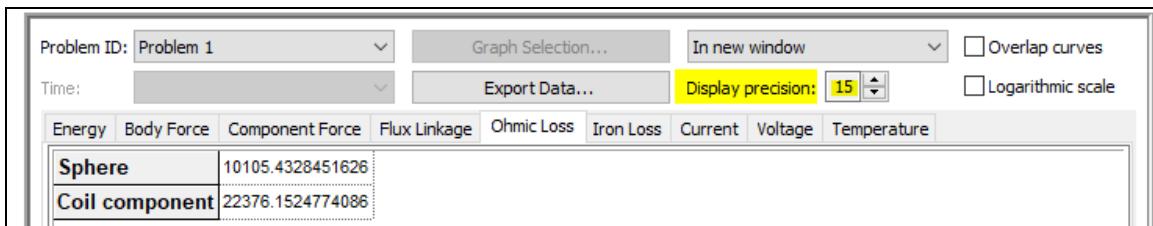
7.2 View the time-averaged ohmic losses

Click the Results tab to switch back to the Results window.

1. Select the *Ohmic Loss* tab.

The *Ohmic Loss* page displays the time-averaged Ohmic loss for each conducting component in the model.

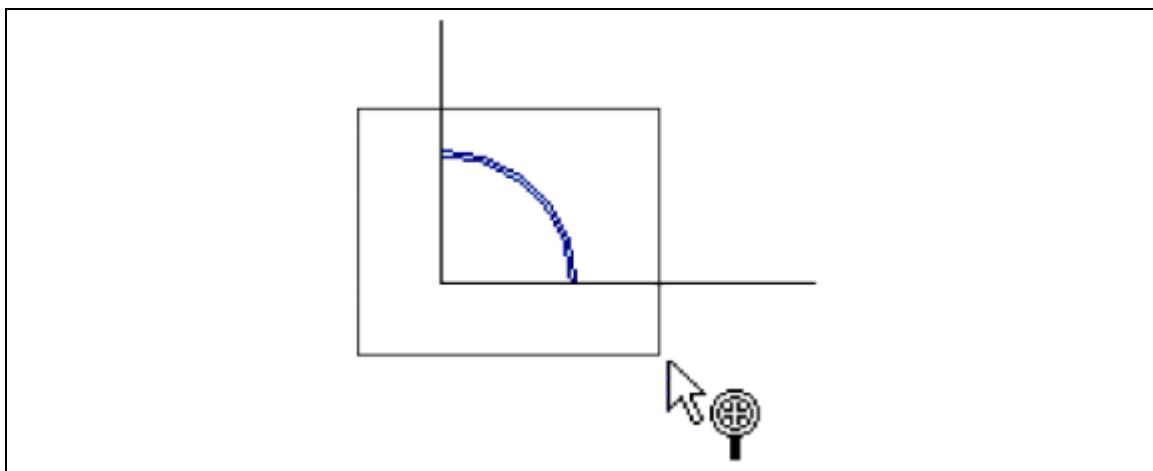
2. If required, using the spin button provided, increase the Display precision to 15.



7.3 Set the color interpolation and style of the shaded plot

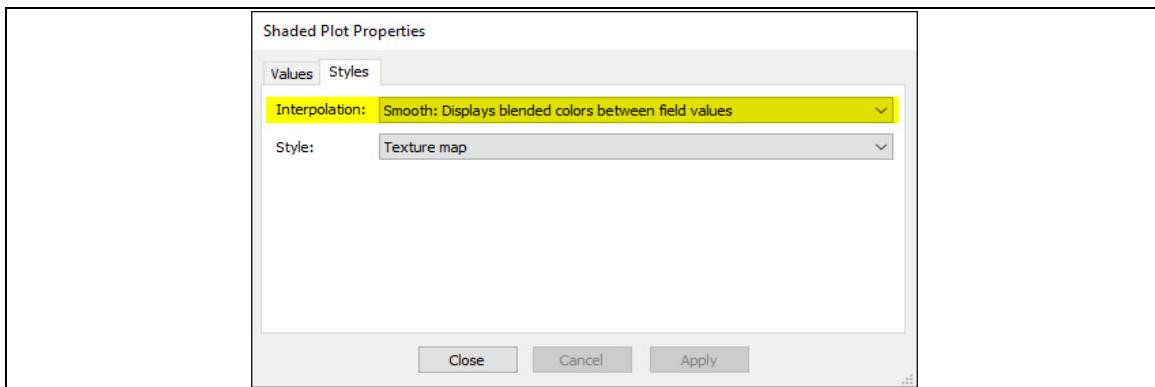
This procedure will set the shaded plots to smooth instead of discrete, which is the default.

1. On the View menu, click *Default Fields*.
2. With the Examine Model tool still active, hold down the CTRL key and the left mouse button to form a rectangular box around the sphere.



3. Release the mouse pointer.
The area enclosed by the rectangle is enlarged.
4. On the Project Bar, select the *View* tab.
5. From the View tree, click *Shaded Plot*.

6. On the Edit menu, click *Properties*.
The Shaded Plot Properties page appears.

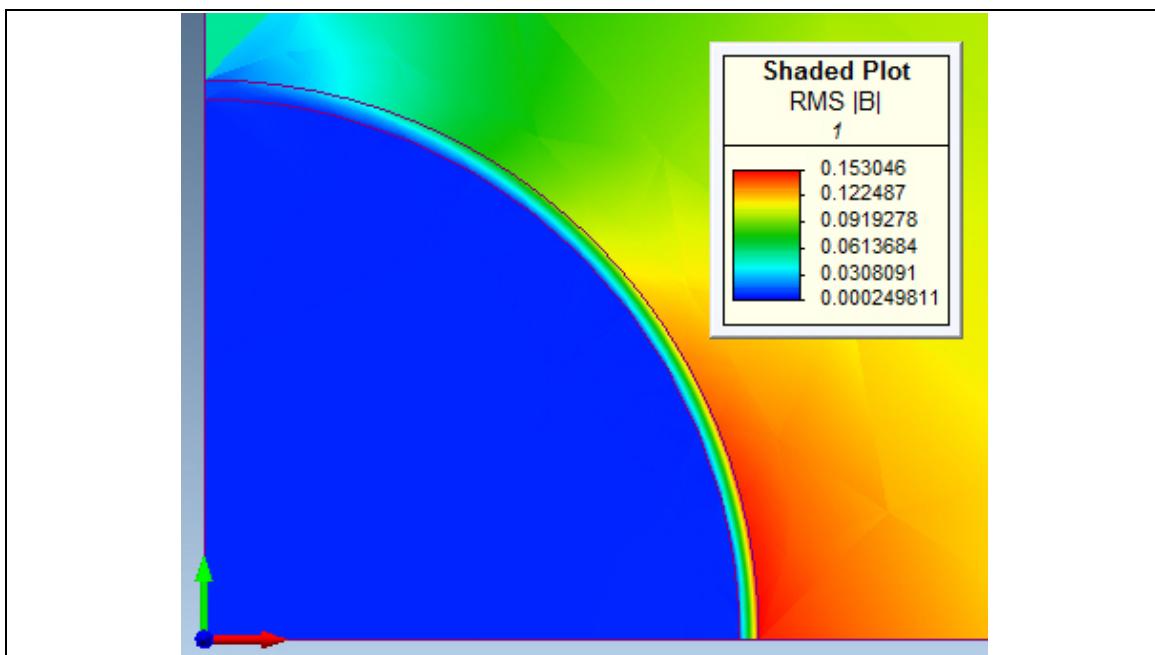


7. Click the Styles tab and then from the Interpolation drop-down menu, select "Smooth: Displays blended colors between field values".
8. Click OK.

7.4 View the shaded plot of RMS |B|

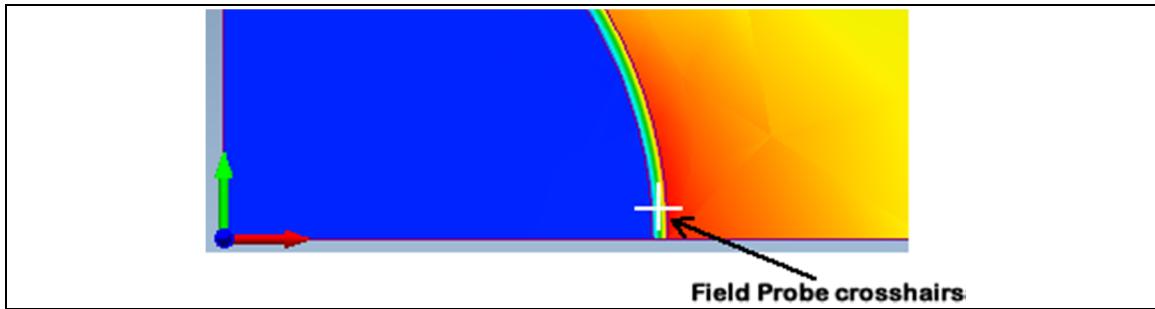
1. On the Project bar, select the *Field* tab.
2. Select the *Contour* tab.
3. In the *Fields to display* list, select **None**.
4. Select the *Shaded* tab.
5. In the *Fields to display* list, select **RMS |B|**.
6. Click Update View.

The shaded plot is displayed with a color legend beside it.



7.5 Probe RMS $|B|$ in the sphere

The Field Probe feature allows you to probe for field values, using the mouse to pinpoint a desired location over the solution. The value is displayed in the Status Bar. You can also write the selected value to the Text Output Bar, which displays the global coordinates of the model along with the value.



7.6 To probe for field values using the mouse

1. On the Tools menu, click *Field Probe* to enable it.
2. Move the mouse (crosshairs) over the solution (as shown in the illustration above).
The field value and its specified location on the solution are displayed in the Status Bar.



3. Click the left mouse button over any area of the solution.

The Text Output Bar automatically opens (if it wasn't already opened), and the x, y, and z coordinates of the location on the solution are displayed along with the field value.

X	Y	Z	Field	Value	Component
0.133804	0.0145302	0	RMS $ B $	0.0898687	Sphere
0.133049	0.0241079	0	RMS $ B $	0.107981	Sphere
0.119468	0.0609452	0	RMS $ B $	0.0690285	Sphere
0.103624	0.0852579	0	RMS $ B $	0.0621124	Sphere
0.0621267	0.120622	0	RMS $ B $	0.0584341	Sphere
0.0259109	0.133146	0	RMS $ B $	0.026063	Sphere

The example above shows the coordinates and $|B|$ field values for several locations that were clicked upon.

7.7 Save the model

You have now completed the time-harmonic version of the Spherical Shield.

1. On the File menu, click *Save*.
2. On the File menu, click *Close*.

8 Summary

In this tutorial, you completed the steps in editing the Spherical Shield model for a time-harmonic solution. The skills you learned include:

- Changing the material of a component
- Setting the source frequency
- Viewing a Contour plot of the solution
- Viewing the time-averaged Ohmic loss in a conductor
- Viewing a Shaded plot of the solution
- Probing a Shaded plot using the Field Probe feature

8.1 Further exploration

In this tutorial, solution accuracy was improved by increasing the mesh density around the sphere. The solution accuracy could be further improved by also raising the polynomial order of the entire model to 3.

Please see the Help for more information on changing the polynomial order.

Tutorial #6

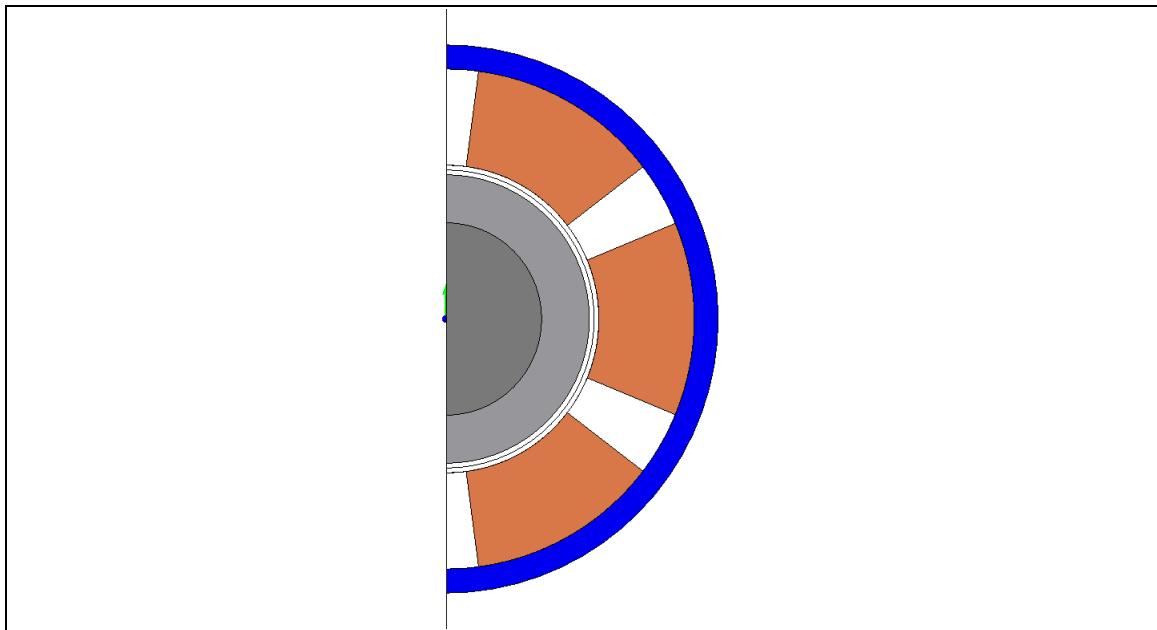
2D Transient with Motion

TEAM Problem 30

1 Modeling plan

The devices described in TEAM problem 30¹ are appropriate applications of the transient solver. One of the devices described in TEAM problem 30 is a 3-phase induction machine. Most of the analyses of this device can be done using the time-harmonic solver. While using the time-harmonic solver is the most economical method and is preferred, there are conditions that require the use of a transient solver. This tutorial intends to examine one or more of these conditions and to demonstrate the steps for setting up a model, solving it, and post-processing the solutions when a transient solver is used.

The following steps guide you through creating a half-model of the 3-phase induction machine and analyzing it in Simcenter MAGNET.



2 Creating a new model

2.1 Opening a new model

- Start Simcenter MAGNET.
The Main window appears.
or
- If Simcenter MAGNET is already running, click *New* on the File menu to open a new model.

2.2 Name the model

Simcenter MAGNET assigns a temporary name to the model (e.g., Simcenter MAGNET 1) every time a new model is opened. As long as the application remains open, each new model number increments by one (e.g., if the new model you have opened is the fourth one in this session, Simcenter MAGNET would assign the temporary name Simcenter MAGNET 4). You give the model a permanent name when you save it.

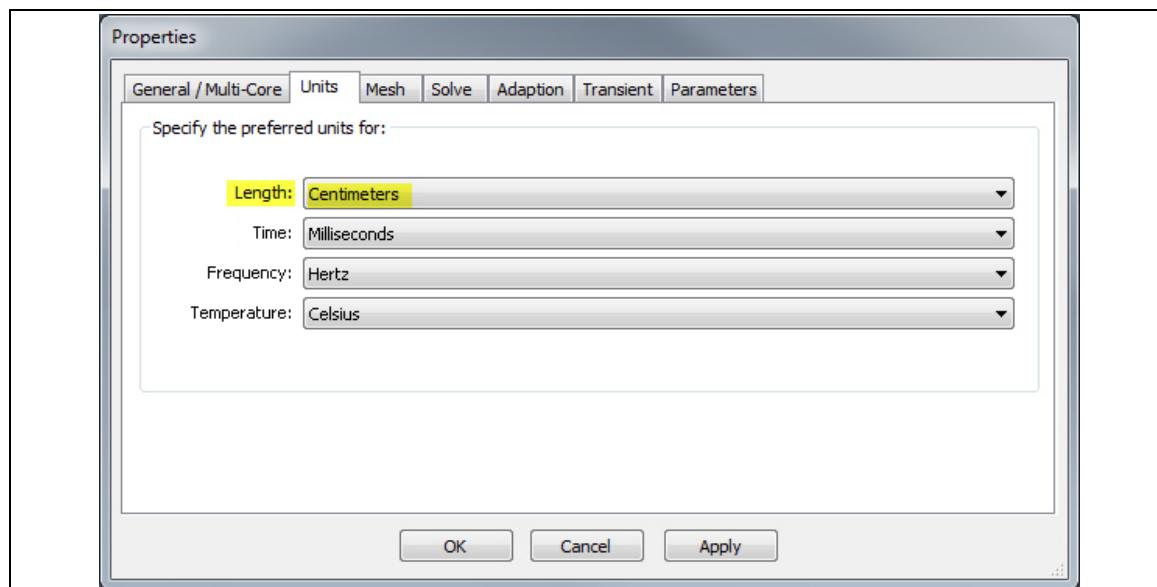
1. On the File menu, click *Save* or *Save As*.
2. In the *Save As* dialog box, type **Team Problem 30** as the name of the model.
3. Choose the drive and directory where you want to place the model.
4. Click *Save As*.

3 Setting up the working environment

3.1 Initial Settings

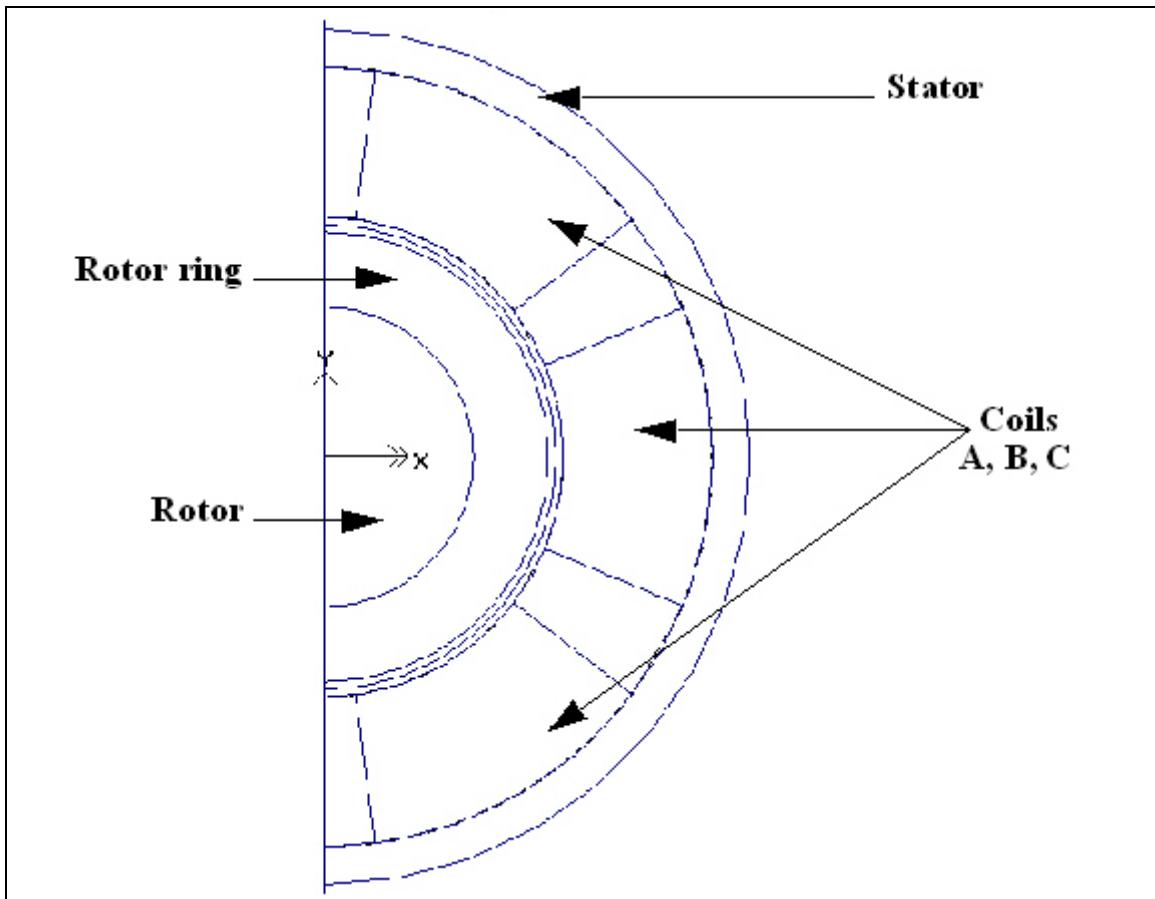
Each new model reverts to the Simcenter MAGNET default settings for the preferred units for length, time, frequency, and temperature, and for how curves will be displayed in the view. For our model, we are going to change only the preferred unit for length, and accept all the other defaults.

1. From the Object page, select the model (i.e. Team Problem 30).
2. On the Edit menu, click *Properties*.
3. Select the *Units* tab.
4. From the Length drop-down list, select **Centimeters**.



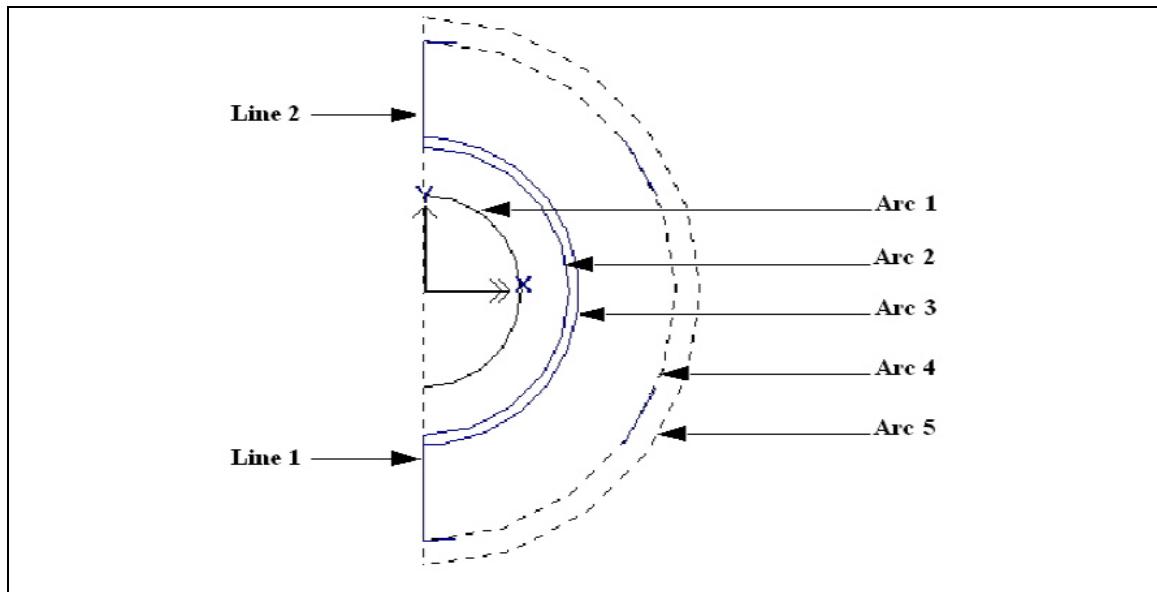
5. Click OK.

- 4 Build the geometric model for the Rotor, Rotor ring, Stator, and Coils, and set up the problem



4.1 Draw the outlines of the rotor, rotor ring, and stator components

The outlines of these components are shown in the illustration below.



In the procedure that follows, only an arc of unit radius is actually drawn. Using the Edge Scaling feature, this arc is then duplicated to create arcs 1 through 5. Lines 1 and 2 are drawn to conclude.

1. On the Tools menu, click *Keyboard Input Bar*.
2. On the View menu, click *Update Automatically*.
Use of this feature resets the view automatically to include the model's entire outline as it is being drawn.
3. On the Draw menu, click *Arc (Center, Start, End)*.
4. In the Keyboard Input bar, enter the following coordinates to draw an arc of unit radius:

Center x, y **0, 0** Press ENTER

Start x, y **0, -1** Press ENTER

End x, y **0, 1** Press ENTER

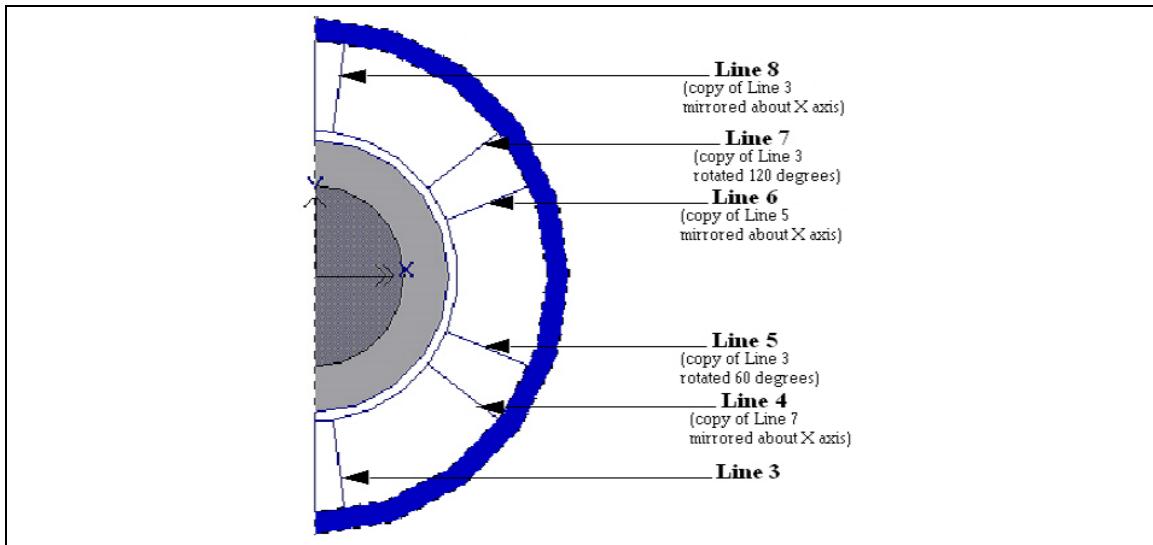
5. On the Edit menu, click *Select Construction Slice Edges*.
6. Using the mouse pointer, select the arc you have just drawn.
7. On the Draw menu, click *Scale Edges*.
8. In the Scale factor text box, type **2**, and select the option *Apply the transformation to a copy of the selection*.
9. Click *Apply*.
Arc 1 is created.
10. Repeat steps 8 and 9 with scale factors of **3, 3.2, 5.2, and 5.7**.
Arcs 2 through 5 are created.
11. Click *Close*.

12. Press *Delete* to remove the arc of unit radius.
13. On the Draw menu, click *Line*.
14. In the Keyboard Input bar, enter the following coordinates to draw lines 1 and 2:

Start x, y	0, -5.7	Press ENTER
End x, y	0, 0	Press ENTER
End x, y	0, 5.7	Press ENTER, and then ESC

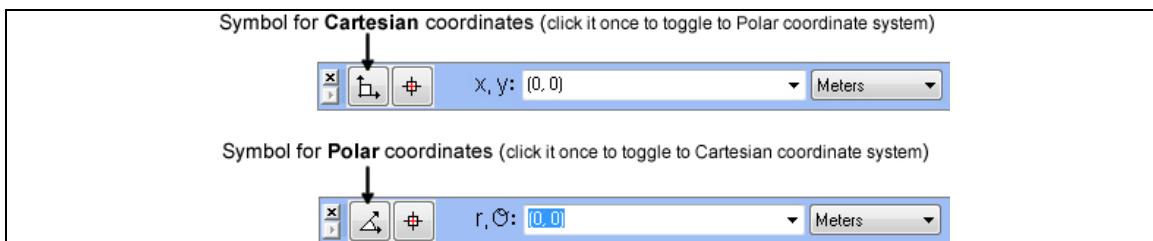
4.2 Draw the outlines of the coil components

The outlines of these components are shown in the illustration below.



In the procedure that follows, only Line 3 is actually drawn. Using the Edge Rotation feature, this line is duplicated to create lines 5 and 7, and using the Edge Mirroring feature, lines 3, 5, and 7 are duplicated to create lines 8, 6, and 4.

1. Toggle the Keyboard Input bar to the Polar coordinate system.



2. In the Keyboard Input bar, enter the following polar coordinates to draw Line 3:

Start r, theta	3.2, -82.5	Press ENTER
End r, theta	5.2, -82.5	Press ENTER, and then ESC

3. On the Edit menu, click *Select Construction Slice Edges*.
4. Using the mouse pointer, select the line you have just drawn.
5. On the Draw menu, click *Rotate Edges*.

6. In the Rotation Angle text box, type **60**, and select the following options:
 - Number of copies → 2
7. Click OK.
Two copies of the line are created: lines 5 and 7.
8. Holding down the CTRL key, use the mouse pointer to click lines 5 and 7 and add them to the selection.
9. On the Draw menu, click *Mirror Edges*.
10. Type (1, 0) as the Direction of the mirror line (the X axis), and select the option *Apply the transformation to a copy of the selection*.
11. Click OK.
Single copies of the lines are created: lines 8, 6, and 4.

4.3 Create new materials

For this problem, you will have to create three new materials in your material database.

1. On the Tools menu, click *New User Material*.
2. On the General page, specify the following:
 - Name: Rotor Steel
 - Display color: Click Display Color and select a dark grey
 - Transparency: *Optional*
 - Description: *Optional*
3. Click *Next*.
4. On the Options page, select the following:
 - Magnetic: Permeability
 - Electric: Conductivity
 - Other: Mass Density
5. Using the Next button to advance to the appropriate pages, enter the following values:

▪ Temperature (Celsius) = 20	Press ENTER
▪ Relative Permeability = 30	Press ENTER
▪ Coercivity (Amps/m) = 0	Press ENTER
▪ Conductivity (Siemens/m) = 1.6e6	Press ENTER
▪ Mass Density (kg/m ³) = 7800	Press ENTER
6. Once you have entered all the values, advance to the Confirmation page and click *Finish* to create the new material.

7. Repeat steps 1 through 6 using the data listed below:

Name: **Rotor Aluminum**

Display color: *Click Set Color and select a medium grey*

Temperature (Celsius) = **20**

Relative Permeability = **1**

Coercivity (Amps/m) = **0**

Conductivity (Siemens/m) = **3.72e7**

Mass density (kg/m³) = **2707**

Name: **Stator Steel**

Display color: *Click Set Color and select a medium blue*

Temperature (Celsius) = **20**

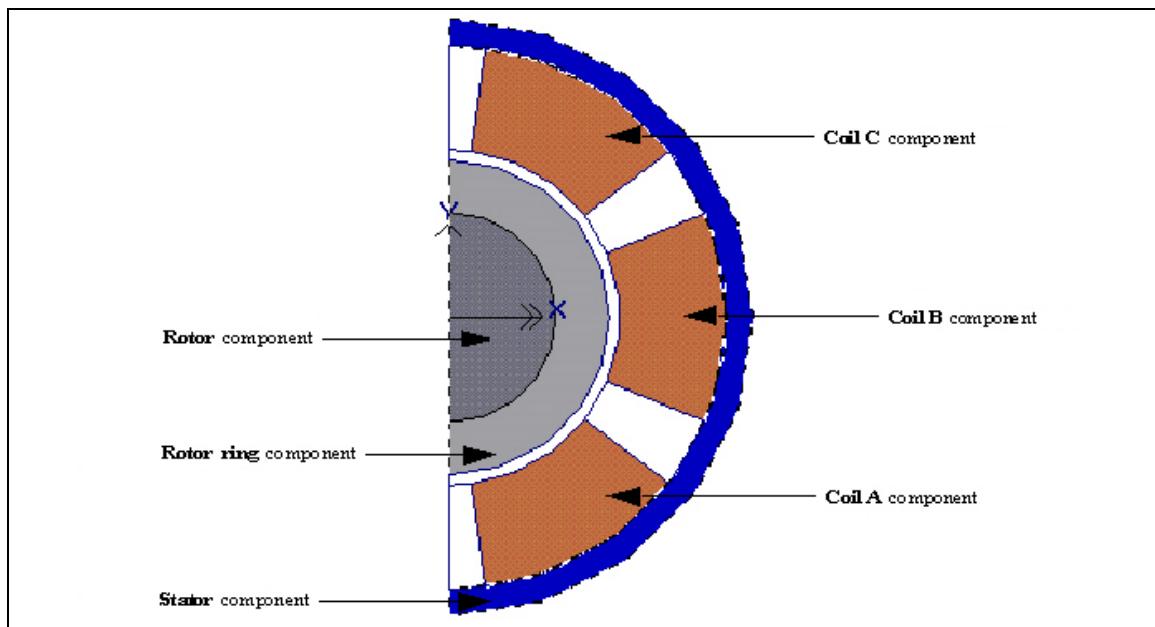
Relative Permeability = **30**

Coercivity (Amps/m) = **0**

Mass density (kg/m³) = **7800**

4.4 Make the rotor, rotor ring, stator, and coil components

The interior regions for these components are shown in the illustration below.

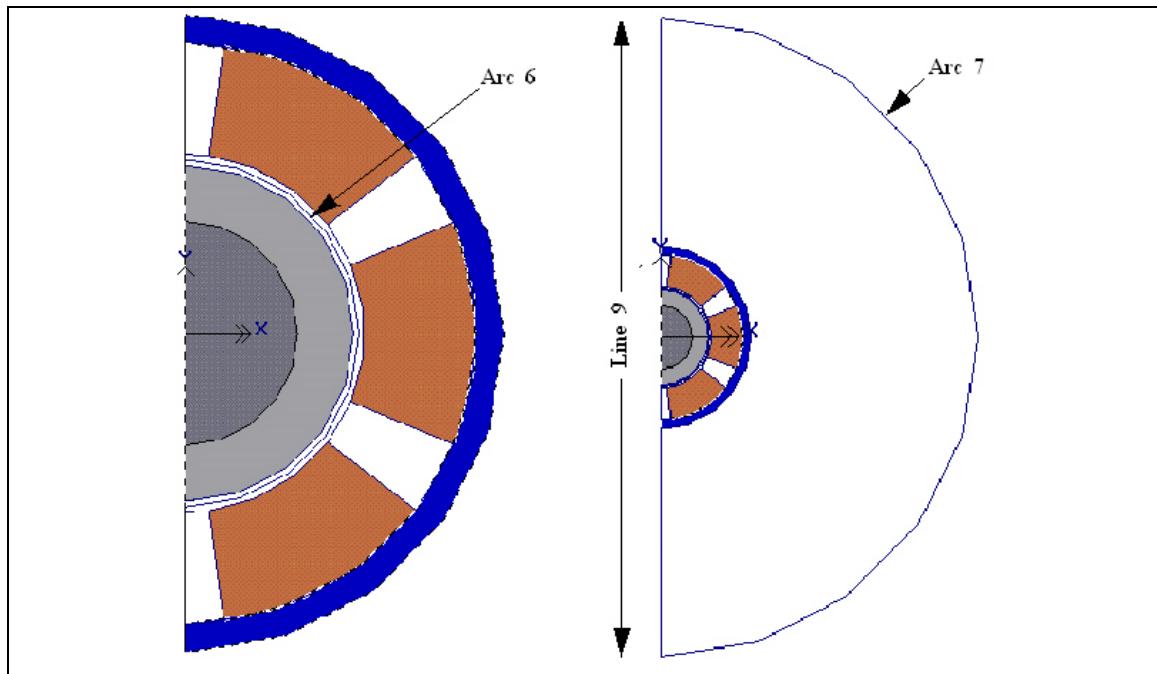


1. On the Edit menu, click Select Construction Slice Surfaces.
2. Select the interior region for **Rotor** from the outlines you have drawn.
3. On the Model menu, click *Make Component in a Line*, and specify the following:
 - Name: **Rotor**
 - Material: **Rotor Steel**
 - Distance: **100 Centimeters**
4. Deselect the option *Remove unnecessary vertices on the boundaries* since the vertex at the origin will be needed when we assign periodic boundary conditions later on in this document.
5. Click OK.
6. Select the interior region for **Rotor ring** from the outlines you have drawn.

7. On the Model menu, click *Make Component in a Line*, and specify the following, keeping the same distance:
 - Name: **Rotor ring**
 - Material: **Rotor Aluminum**
8. Reset the option *Remove unnecessary vertices on the boundaries*. We will leave it selected for all other components.
9. Click OK.
10. Select the interior region for **Stator** from the outlines you have drawn.
11. On the Model menu, click *Make Component in a Line*, and specify the following:
 - Name: **Stator**
 - Material: **Stator Steel**
12. Click OK.
13. Select the interior region for **Coil A** from the outlines you have drawn.
14. On the Model menu, click *Make Component in a Line*, and specify the following:
 - Name: **Coil A**
 - Material: **Copper: 5.77e7 Siemens/meter**
15. Click OK.
16. Repeat steps 13 through 15 to make **Coil B** and **Coil C**.
17. On the File menu, click *Save*.

4.5 Draw the outlines of the air components

The outlines of these components are shown in the illustration below.



For this type of problem where the periodic boundary condition is used, it is necessary to divide the air gap between the rotor and the stator. An air box surrounding the model is also required to model the field leaking outside the device. This field will affect the results since the stator has an unusually low permeability and is very thin.

1. On the Draw menu, click *Arc (Center, Start, End)*.
2. Toggle the Keyboard Input bar to the Cartesian coordinate system.
3. In the Keyboard Input bar, enter the following coordinates to draw arcs 6 and 7:

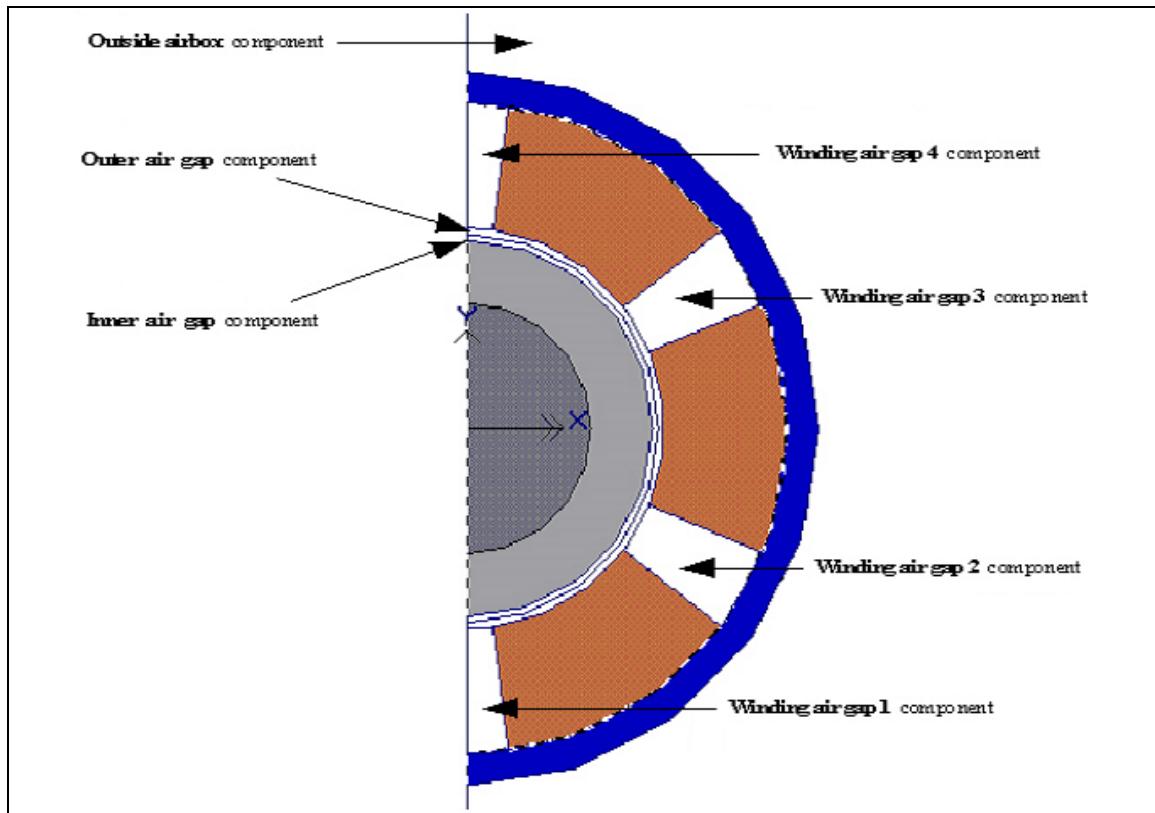
Center x, y	0, 0	Press ENTER
Start x, y	0, -3.1	Press ENTER
End x, y	0, 3.1	Press ENTER
Center x, y	0, 0	Press ENTER
Start x, y	0, -20	Press ENTER
End x, y	0, 20	Press ENTER

4. On the Draw menu, click *Line*.
5. In the Keyboard Input bar, enter the following coordinates to draw Line 9:

Start x, y	0, 20	Press ENTER
End x, y	0, -20	Press ENTER, and then ESC

4.6 Make the air components

The interior regions for these components are shown in the illustration below.



1. On the View toolbar, click (Examine Model).
2. Hold down the CTRL key and the left mouse button to form a rectangular box around the regions for the Inner and Outer air gaps.
3. On the Edit menu, click Select Construction Slice Surfaces.
4. Select the interior region for **Inner air gap** from the outlines you have drawn.
5. On the Model menu, click *Make Component in a Line*, and specify the following:
 - Name: Inner air gap
 - Material: AIR
6. Click OK.
7. Repeat steps 4 through 6 to make **Outer air gap**.
8. On the View toolbar, click (Examine Model).
9. Roll the mouse wheel down until the regions of the model that you see are similar to what is shown in the illustration above.
10. On the Edit menu, click Select Construction Slice Surfaces.
11. Repeat steps 4 through 6 to make **Winding air gap 1 to 4** and **Outside airbox**.

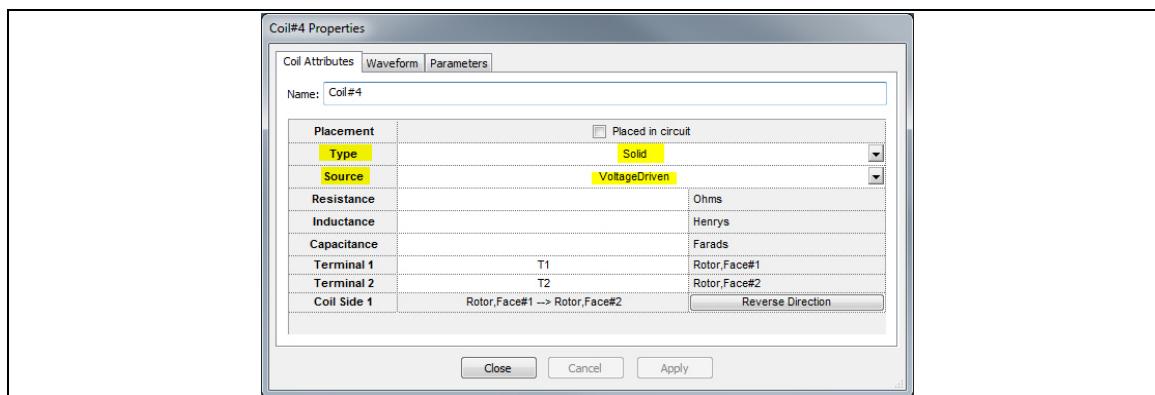
4.7 Make coils

1. From the Object page, select the Coil A component.
2. On the Model menu, click *Make Simple Coil*.
Coil#1 is created and listed in the Object page.
3. Repeating steps 1 and 2:
 - Select **Coil B** and create **Coil#2**
 - Select **Coil C** and create **Coil#3**
 - Select **Rotor** and create **Coil#4**
 - Select **Rotor ring** and create **Coil#5**

Note The reasoning behind making coils out of the **Rotor** and **Rotor ring** components is that only their halves are modeled. The periodicity being odd, we would expect no net current in either full component, but some net current in each halved component. To allow for that, a coil must be made from each half-component so that its ends can be electrically shorted. This would not be necessary if the entire device were modeled.

4. From the Object page, select **Coil#4**.
5. On the Edit menu, click *Properties*.
The Coil Properties dialog appears.
6. In the Coil Attributes page, do the following:

- for Type, select **Solid**
- for Source, select **VoltageDriven**



7. Click *Apply*.
8. Keeping the *Coil property* dialog open, select Coil#5 from the Object page.
9. Repeat step 6.
10. Click OK.

4.8 Delete the lines on the construction slice

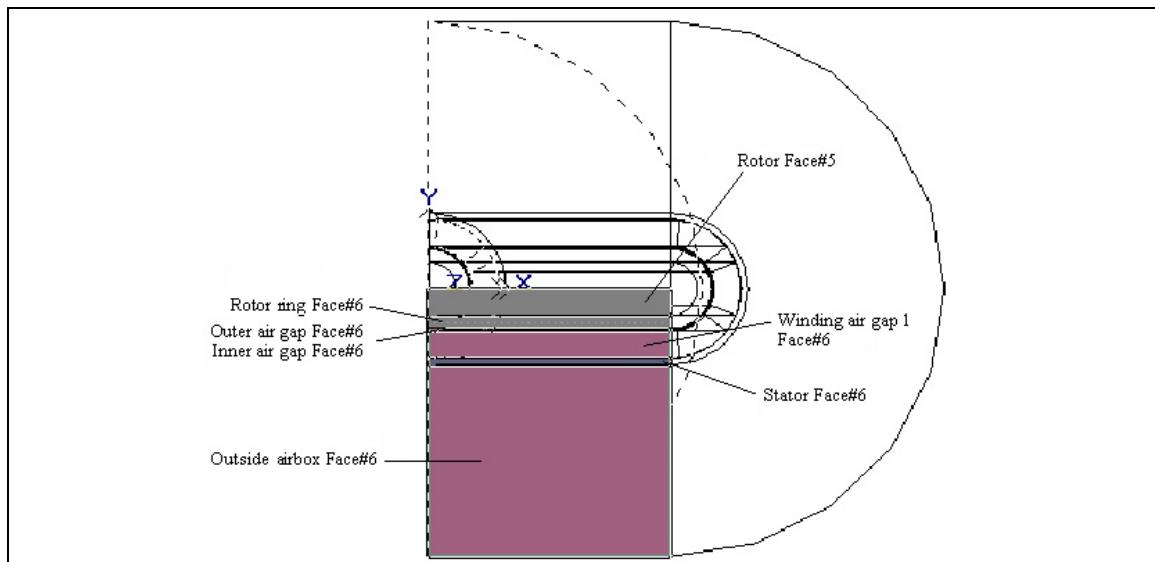
We no longer need the construction slice lines that were used to create the components, so we will proceed to remove them.

1. On the Edit menu, click *Select Construction Slice Edges*.
2. On the Edit menu, click *Select All*.
3. On the Edit menu, click *Delete*.

4.9 Assign boundary conditions

A given side of the half model is equivalent to the other side as seen from the omitted half. Therefore, we need to assign periodic boundary conditions to force the mesh nodes on both sides of the half model to match.

1. On the View menu, click *Wireframe Model*.
2. On the View menu, click *Update Automatically*.
3. On the View menu, click *Rotate → Rotate Right*.
4. Right-click on the Object tab and select *Expand All* to display all the faces in the Object page.
5. Holding down the CTRL key, use the mouse pointer to click and select the following surfaces from the Object page:
 - Rotor,Face#5
 - Rotor ring,Face#6
 - Stator,Face#6
 - Inner air gap,Face#6
 - Outer air gap,Face#6
 - Winding air gap 1,Face#6
 - Outside airbox,Face#6

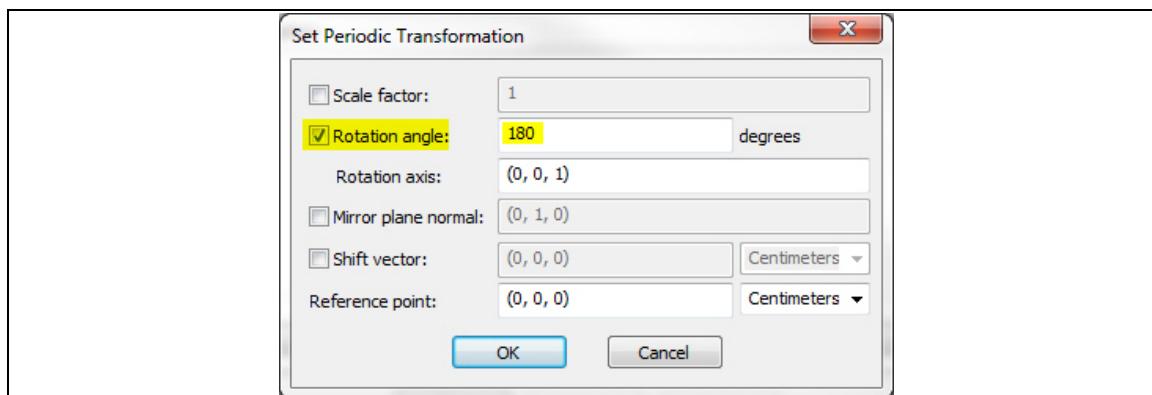


6. On the Boundary menu, click *Odd Periodic*.

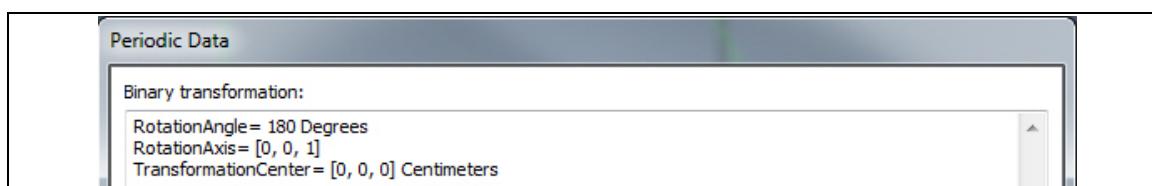
Note Odd Periodic is selected because our device has an odd number of poles.

7. At the bottom of the Periodic Data dialog, click *Set Transformation*.

8. In the Set Periodic Transformation dialog, click the Rotation angle check box, type **180** in the text box, and click OK.

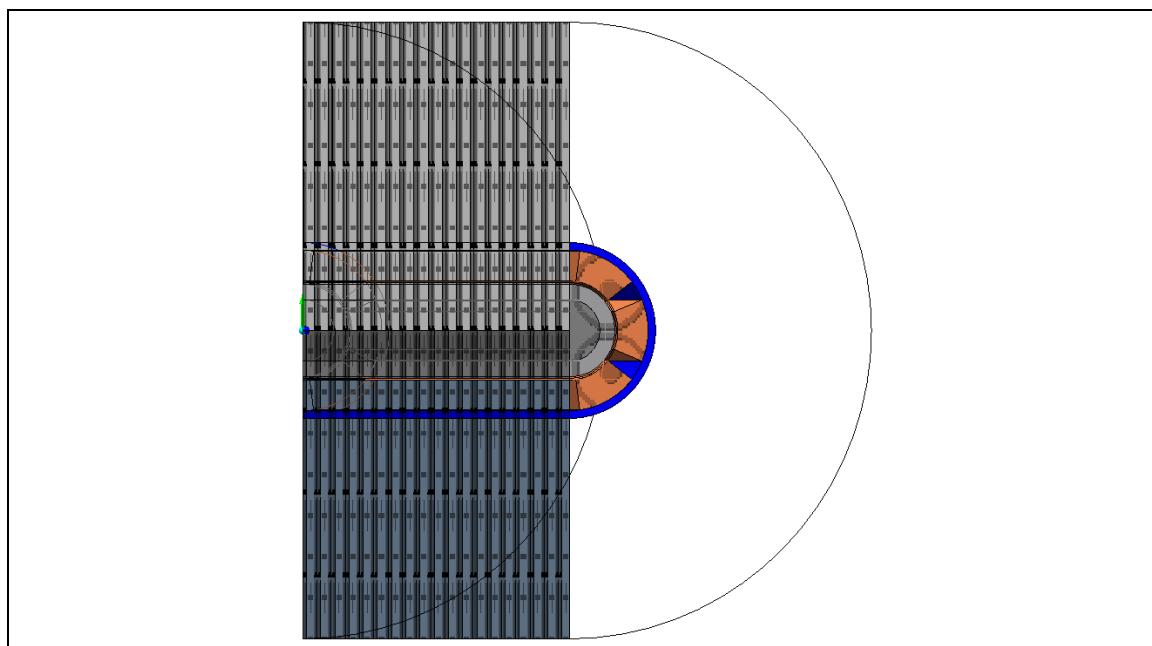


9. Verify that the *Binary transformation* information in the Periodic Data dialog is correct, and then click OK.



10. On the View menu, click *Solid Model*.

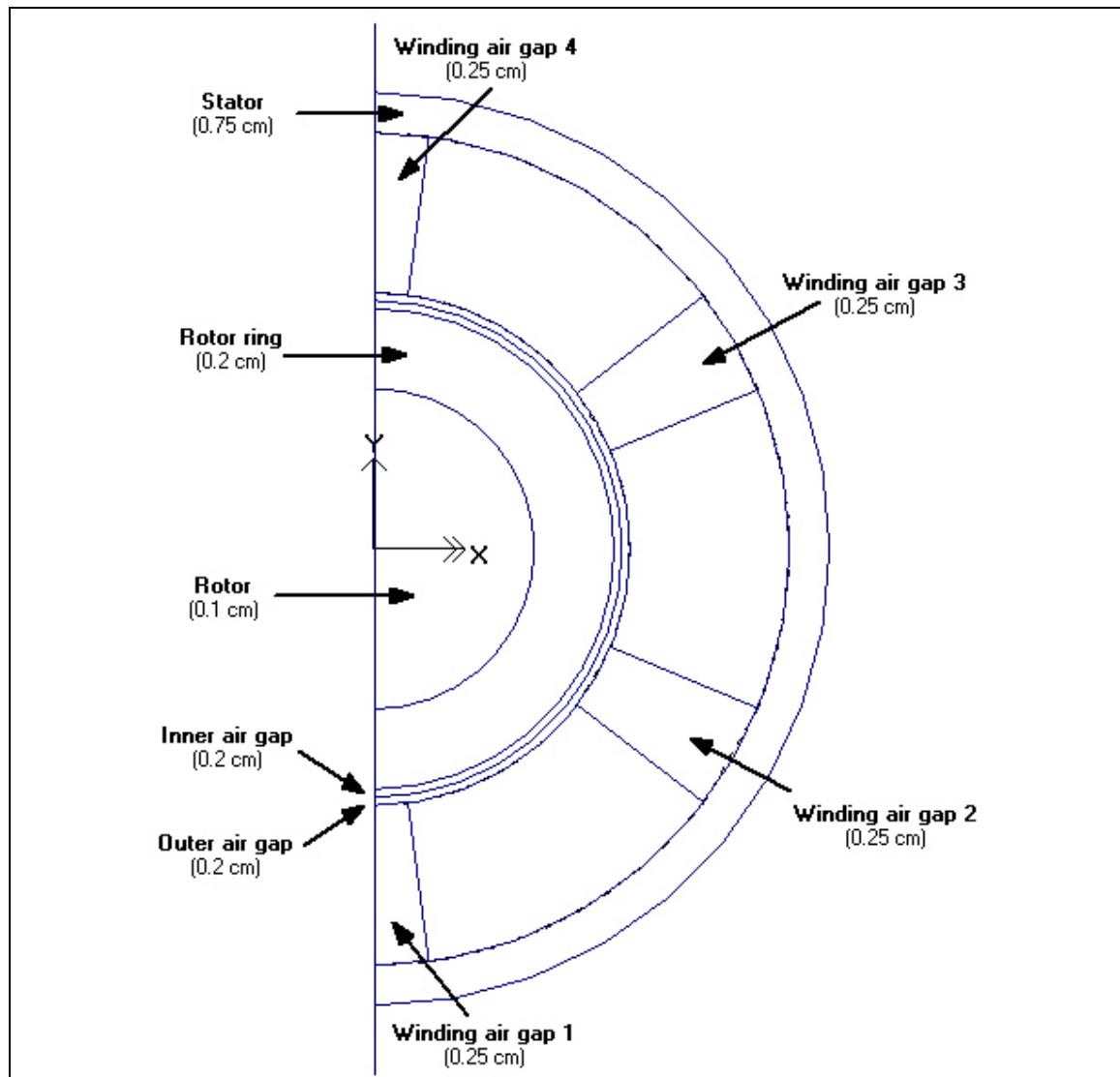
Verify that the periodic conditions have been applied properly to your model. It should look similar to the illustration below.



4.10 Modify the mesh

In the 2D finite element method of analysis, the solution domain is divided into a mesh of triangular elements. The accuracy of the solution depends upon the nature of the field and the size of the mesh elements. In regions where the direction or magnitude of the field is changing rapidly, high accuracy requires smaller elements. One method of increasing the mesh density is to set the maximum element size for a component volume or specific faces of a component. The following procedures will demonstrate this method.

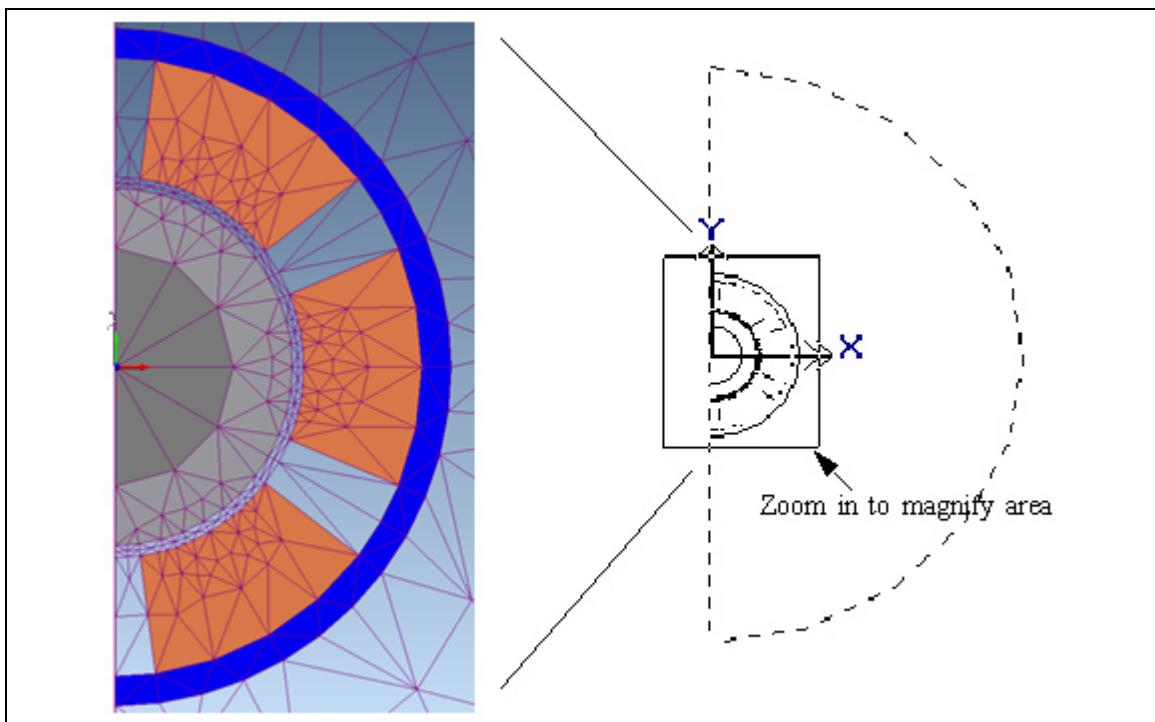
The diagram below shows (in parentheses) the Maximum element size modifications that will be applied to each component.



4.11 View the initial mesh

Before changing the maximum element sizes, the default initial mesh can be viewed.

1. On the View menu, click Preset Views → Positive Z Axis.
2. On the View toolbar, click  (Examine Model).
3. Hold down the CTRL key and the left mouse button to form a rectangular box around the model, excluding the Outside airbox (as shown to the right of the below illustration).



4. On the View menu, click *Initial 2D Mesh*.

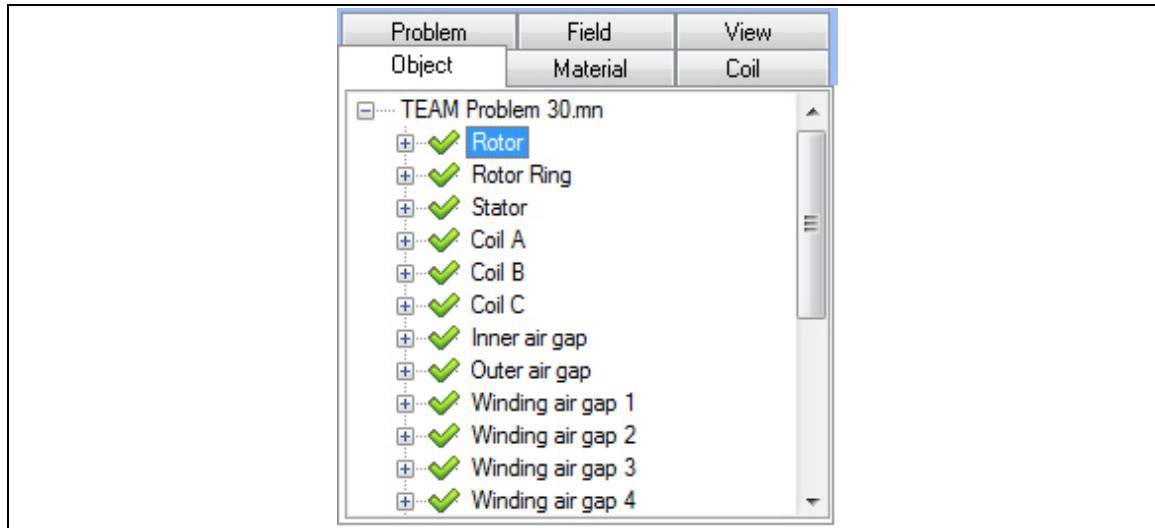
The initial mesh appears in the View window. The initial mesh is displayed on the XY plane, at $Z=0$.

The mesh should look like the left side of the illustration shown above.

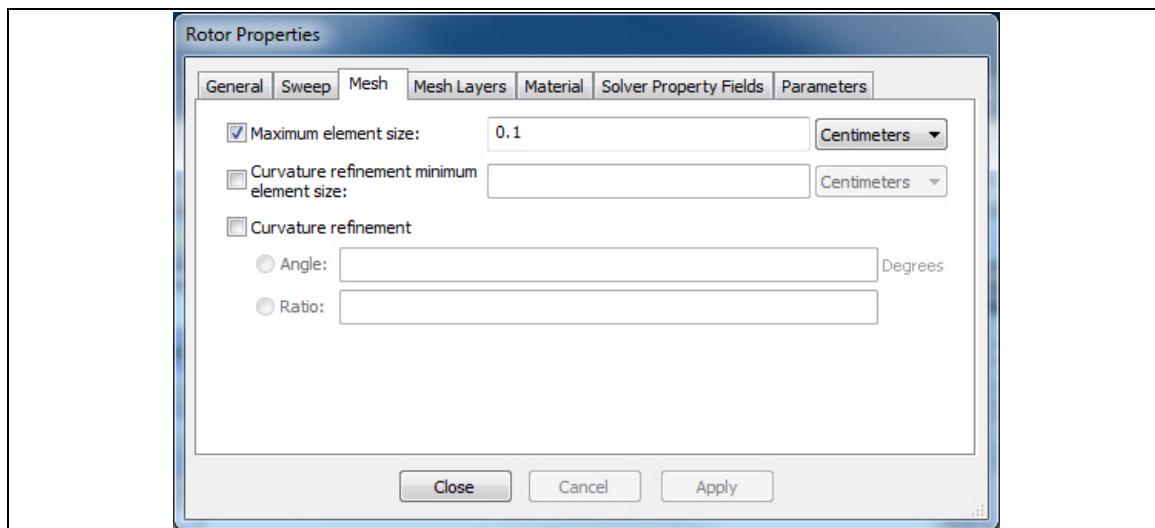
5. On the View menu, click *Solid Model*.

4.12 Set the maximum element size for each component

1. Right-click on the Object tab and select *Collapse All* to hide all the faces in the Object page.
2. In the Object page, select the Rotor component.



3. On the Edit menu, click *Properties*.
The Properties dialog appears.
4. Select the *Mesh* tab.
5. Click the *Maximum element size* checkbox, and type **0.1** in the text box.



6. Click *Apply*.

Tip Clicking *Apply*, instead of *OK*, keeps the dialog open and allows us to proceed to the next component without having to repeat step 2.

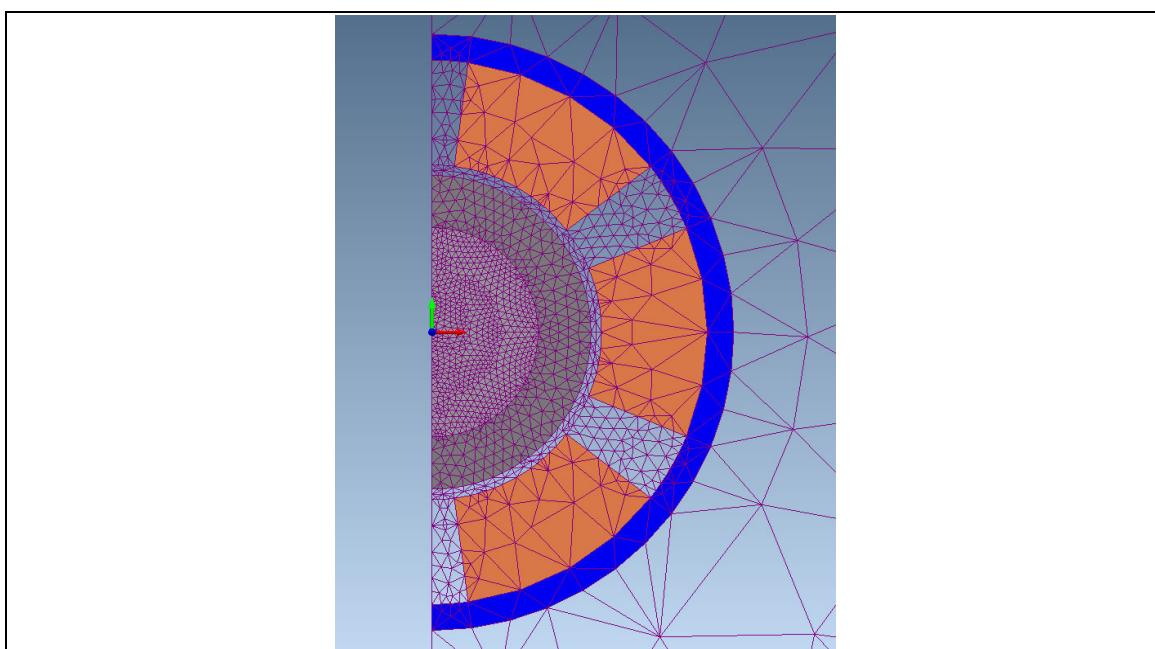
7. In the Object page, select the Rotor ring component.
Notice that the Properties dialog is now titled *Rotor ring Properties*.
8. Click the *Maximum element size* checkbox, and type **0.2** in the text box.
9. Click *Apply*.
10. Repeat steps 7 through 9 with the following components and maximum element sizes:

Component	Maximum element size
Stator	0.75 cm
Inner air gap	0.2 cm
Outer air gap	0.2 cm
Winding air gap 1	0.25 cm
Winding air gap 2	0.25 cm
Winding air gap 3	0.25 cm
Winding air gap 4	0.25 cm

11. Click Close.

4.13 View the changes to the mesh

1. In the Object bar, click on the model name to deselect all components.
2. On the View menu, click *Initial 2D Mesh*.
The initial mesh with the *maximum element size* modifications appears in the View window and should look like the illustration shown below.



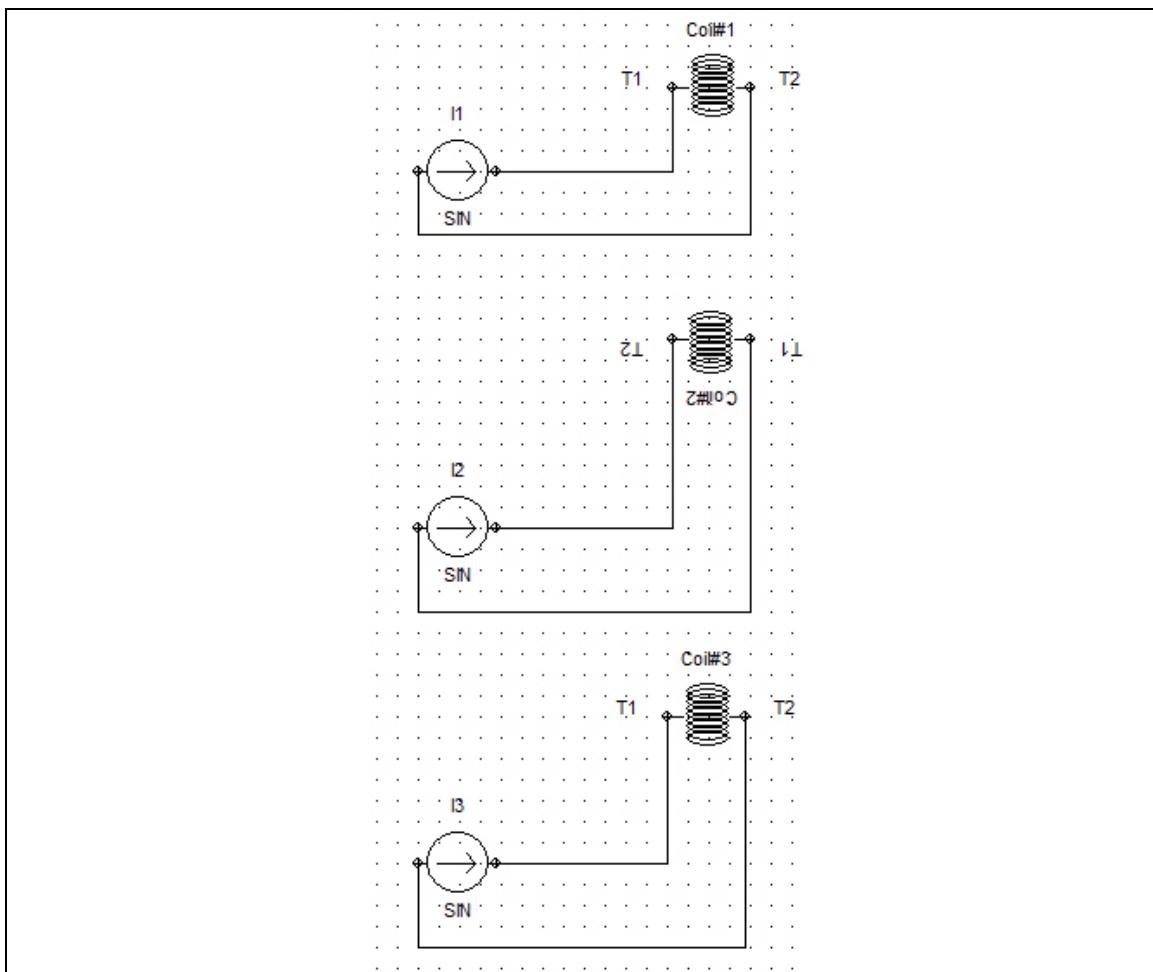
3. On the View menu, click *Solid Model*.

5 Setting up the problem for locked-rotor analysis

In locked-rotor analysis, the torque is measured with the rotor in a locked position. This condition can be analyzed using the time-harmonic solver.

5.1 Creating the circuit for locked-rotor analysis

Before doing this analysis, the driving circuitry should be defined. This is accomplished by creating the following three circuits.



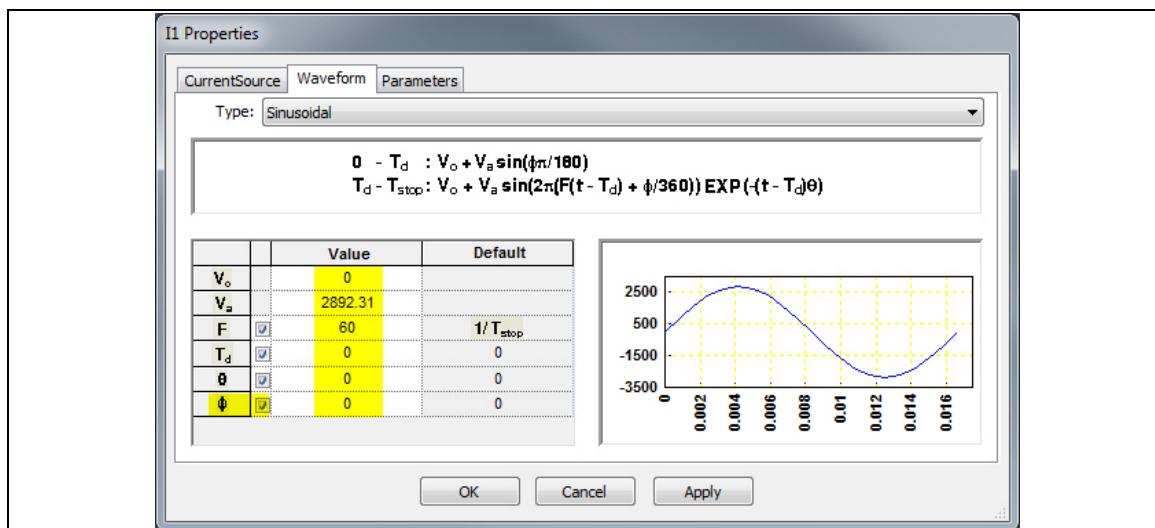
1. On the Window menu, click *New Circuit Window*.
The Circuit window opens with Coil#1 through Coil#5 appearing in the top left frame.
2. Place the cursor over Coil#1, and keeping the mouse button clicked, drag the coil to the Circuit window. Repeat this step for Coil#2 and Coil#3, placing them as shown in the illustration above.
3. Click and select Coil#2 in the Circuit window.
4. On the View menu, click *Rotate* → *Rotate Clockwise*. Repeat this step once more so that Coil#2 is rotated 180 degrees.
5. On the Circuit menu, click *Current Source*.

6. In the Circuit window, click down and to the left of each coil to create the current source components I1, I2, and I3.
7. On the Edit menu, click *Select*.
8. Holding down the CTRL key, click and select I1, I2, and I3.
9. On the Circuit menu, click *Align Components* → *Align Center*.
10. Holding down the CTRL key, click and select Coil#1, Coil#2, and Coil#3.
11. On the Circuit menu, click *Align Components* → *Align Center*.
12. On the Circuit menu, click *Connection*
13. Make connections between terminals according to the pattern shown on page 19. For example, to connect I1 – Left Terminal to Coil#1 – T2:
 - Click on I1 – Left Terminal
 - Click 3 grid points below, where the connection angles to the right
 - Click further right, where the connection angles up toward Coil#1 – T2
 - Click on Coil#1 – T2

Note The connections must be made between terminals. It is impossible to connect a terminal to a wire, or a wire to another wire.

5.2 Modifying the current source components

1. In the Object page, select **I1**.
2. On the Edit menu, click *Properties*.
The Current Source (I1) Properties dialog appears.
3. Select the *Waveform* tab.
4. In the Type drop-down list, select *Sinusoidal*.
5. Click the Φ checkbox to enable *Phase in degrees* and all preceding optional values.
6. Type in the numbers highlighted in the figure below.



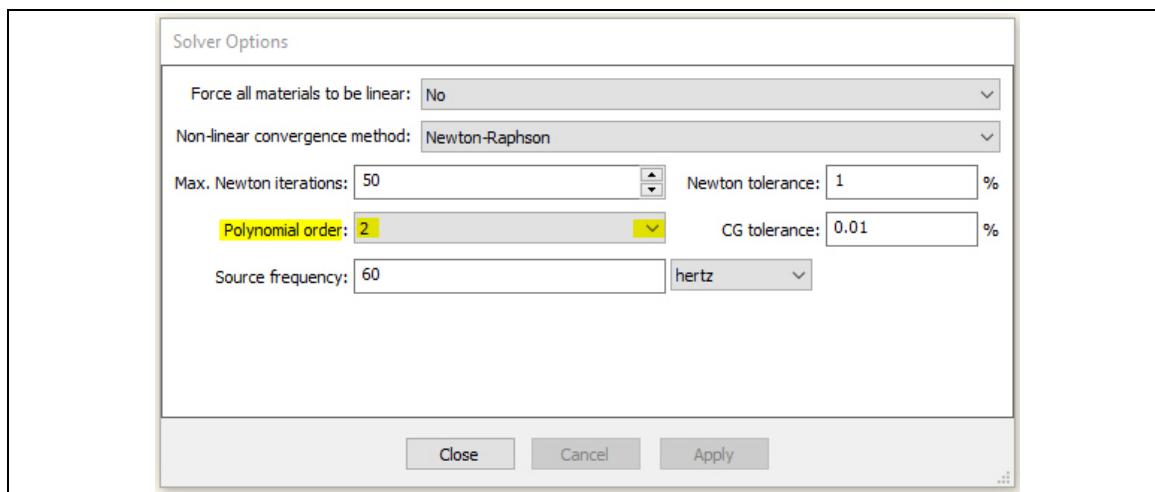
7. Click the *Value* header to select the six data cells, and copy them to the clipboard using CTRL-C.
8. Click *Apply*.

9. In the Object page, select **I2**.
10. Click the Φ checkbox to enable *Phase in degrees* and all preceding optional values.
11. Click the *Value* header, paste the numbers from the clipboard using Ctrl-V, and change Φ to 120 degrees.
12. Click **Apply**.
13. Repeats steps 9 to 11 for **I3**, changing Φ to 240 degrees.
14. Click **OK**.
15. On the File menu, click *Save*.
16. Close  the Circuit window.

6 Generating the Time-Harmonic field solution

6.1 Set the solving options

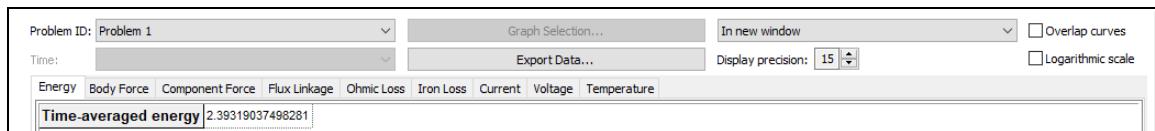
1. On the Solve menu, click *Set Solver Options*.
The *Solver Options* dialog appears.
2. Select **2** as the Polynomial order value, and accept all other default values, as shown in the illustration below.



3. Click **OK**.

6.2 Solve the model

- On the Solve menu, click *Time-Harmonic 2D*.
The Time-Harmonic 2D Solver Progress dialog appears briefly, and then the Results window opens.



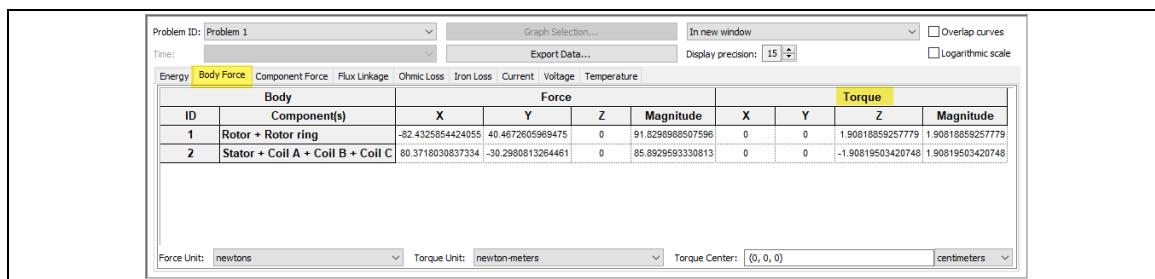
7 Analyzing the results for locked-rotor analysis

In this section, we will examine the Results window and check the value of the torque for the body that is made of the rotor and rotor ring – this is the locked-rotor torque. The theoretical result is 3.826 Nm for a full model. In this tutorial, since we have only modeled half of the device, the torque that Simcenter MAGNET calculates should be closer to half of that value, i.e. 1.908 Nm. This is the case as you can see below.

7.1 View the torque

To view the torque, open the Body Force page.

- Verify that the calculated value is 1.908 Nm.

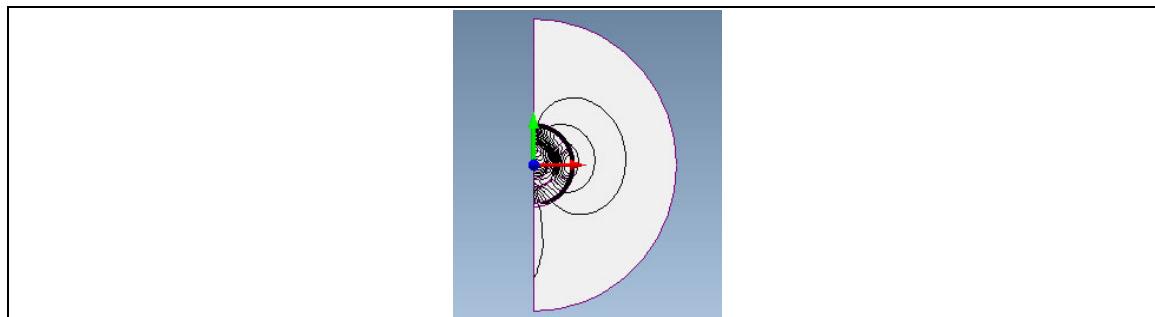


7.2 View the magnetic flux lines

The magnetic flux lines are the contour plot of the Flux function field.

- Before viewing the contour plot, switch back to the View window by clicking the View tab **View 1** located at the bottom of the window.
- On the View menu, click *Update Automatically*.
- On the Project bar, select the Field tab.
- In the *Fields to display* list of the Contour page, make sure that **Flux Function at 0°** is selected.
- Select the *Shaded* tab.
- At the top of the *Fields to display* list, select **None**.
- At the bottom of the Field page, click *Update View*.

The contour plot should look similar to the illustration shown below.



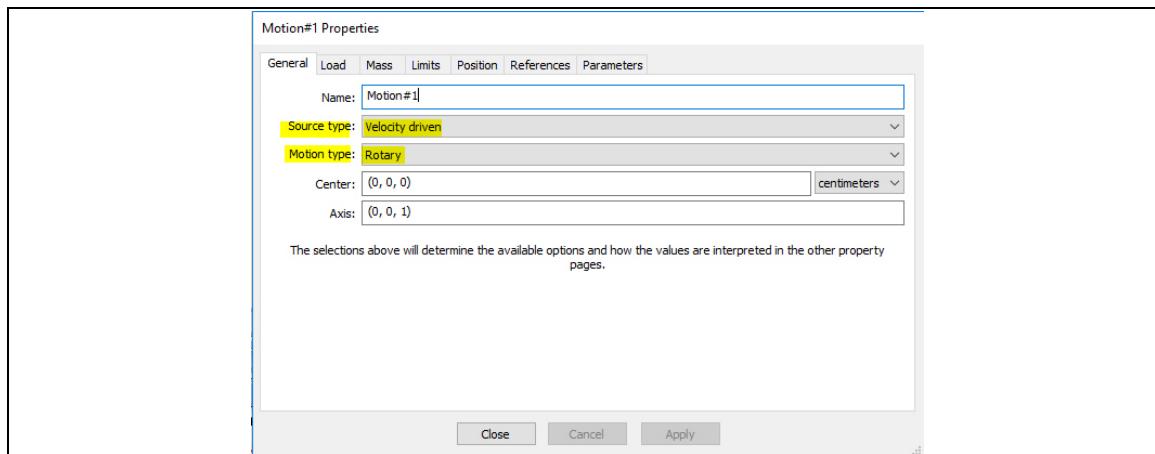
8 Setting up the problem for analysis of torque at a speed of 200 rad/s

Whenever the velocity is constant, it is possible to use the time-harmonic solver by adjusting the frequency of the supply to be the slip frequency. For this part of the tutorial, however, the 2D transient solver is used to solve the problem when the rotor has a constant velocity of 200 rad/s. As a result, a full transient analysis is provided from time zero to when the operation of the induction machine has reached steady-state.

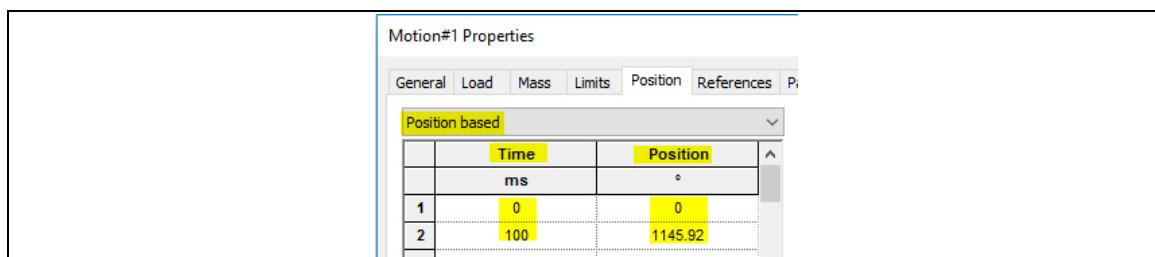
8.1 Making the motion component

For this procedure we will select the rotor, rotor ring, and inner air gap components to create a motion component. The motion component is set to be velocity-driven with rotary motion type. We will also set the velocity by specifying values for the position as a function of time.

1. On the View menu, click *Solid Model*.
2. In the Object page, holding down the CTRL key, click on the **Rotor**, **Rotor ring**, and **Inner air gap** components.
All three components are selected.
3. On the Model menu, click *Make Motion Component*.
The Motion Component dialog appears.
4. In the General page, set the Source type as **Velocity driven** and the Motion type as **Rotary**.



5. Select the Position tab, select *Position based* from the drop-down list, and then enter the data shown below:



6. Click OK.

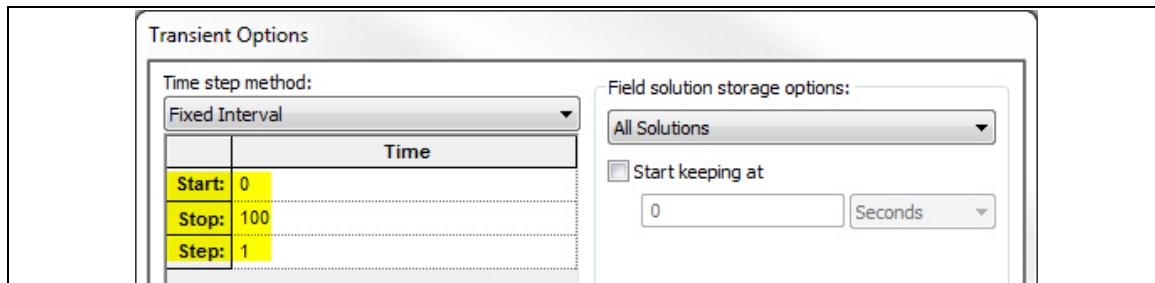
9 Generating the Transient 2D with Motion field solution

9.1 Set the Transient options

1. On the Solve menu, click *Set Transient Options*.

The Transient Options dialog appears.

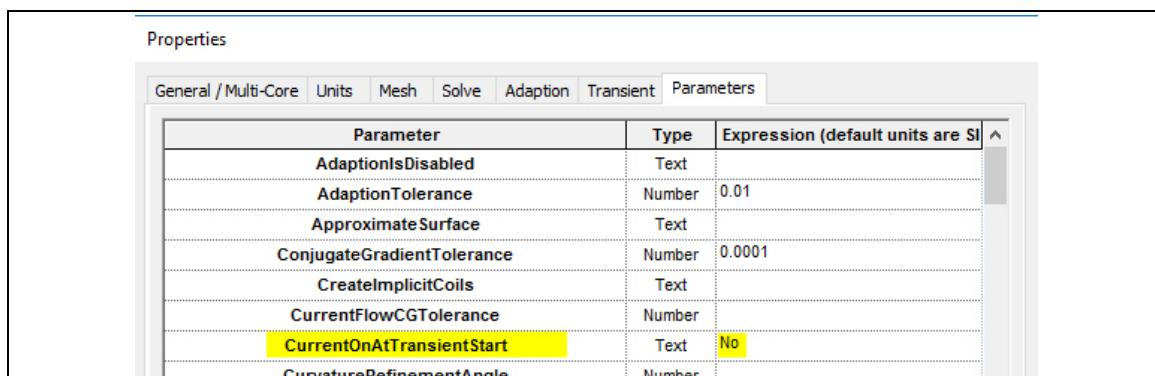
2. Make sure that Fixed Interval is selected as the Time step method, and enter in the Time column the values shown below:



3. Click OK.

9.2 Specifying how the current source waveforms are interpreted when the Transient solver is started

1. In the Object page, select **TEAM Problem 30.mn**.
2. On the Edit menu, click *Properties*.
The Model Properties dialog appears.
3. Select the *Parameters* tab.
4. Scroll down to *CurrentOnAtTransientStart* and type No.



5. Click OK.

9.3 Solve the model

- On the Solve menu, click *Transient 2D with Motion*.

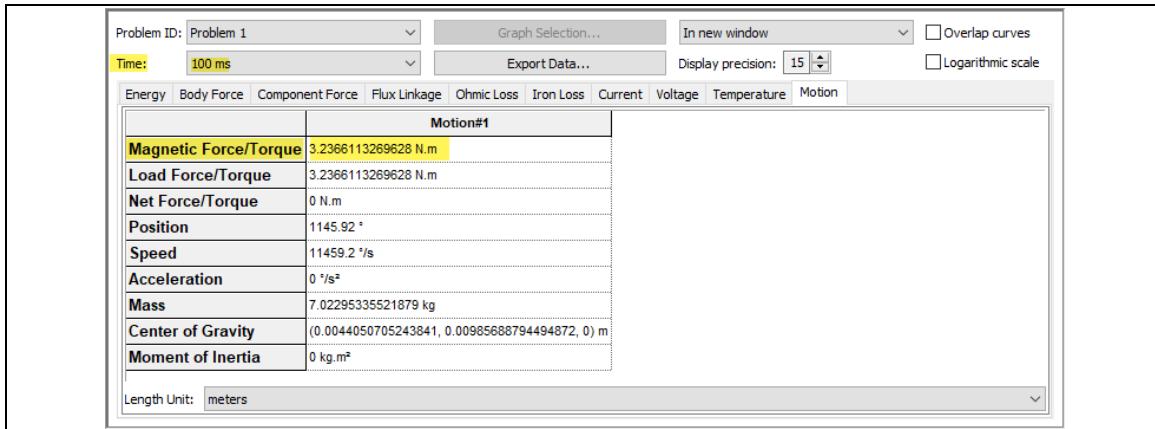
The Transient 2D with Motion Solver Progress dialog appears until the solution is complete, and then the Results window is automatically displayed.

10 Analyzing the results for torque at a speed of 200 rad/s

In this section, we will examine results for the motion component by selecting the entry for magnetic torque and graphing the result. A graph is produced that displays the torque for the rotor when it is rotating at 200 rad/s. The value of torque at steady state should be 3.253 Nm for the half model.

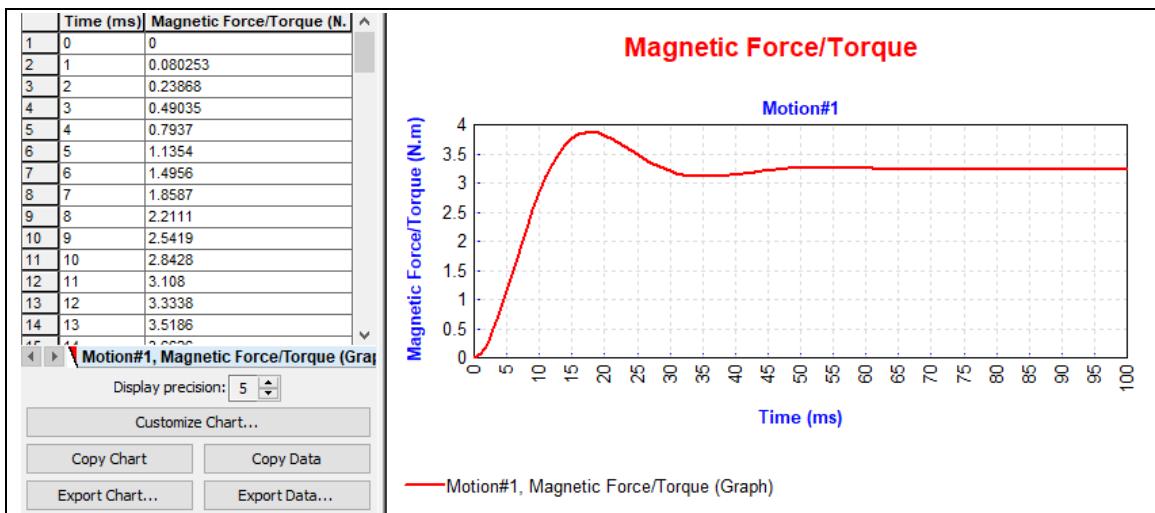
10.1 Graph the magnetic torque

1. In the Results window, open the *Motion* page.
2. Select *100 ms* from the *Time* drop-down list.



3. Using the mouse pointer, click anywhere inside the Magnetic Force/Torque data box (i.e. 3.2366113269628 N.m).
4. Click the *Graph Selection* button, located at the top of the Results window.

The graph should look like the illustration below.



5. On the File menu, click *Save*.

11 Setting up the problem for analysis of the start-up condition

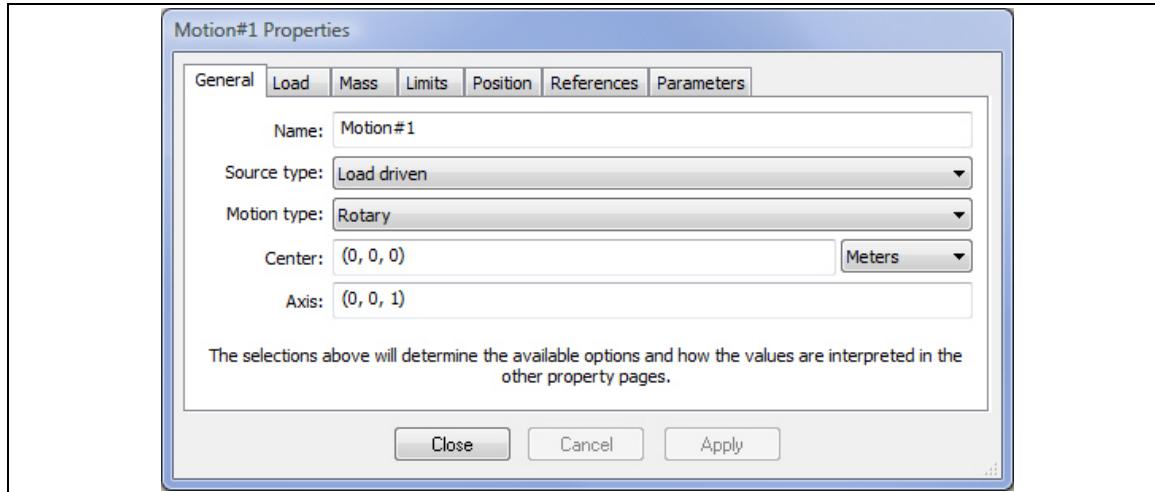
In this analysis of the start-up condition, we will examine the rotor speed and torque as a function of time.

11.1 Modifying the motion component

For the purpose of this analysis, we will need to set the motion component to be load-driven. This means that the velocity of the motion component is not known and that it varies with time as a result of the generated electromagnetic field.

1. In the Object page, select the Motion#1 component.
2. On the Edit menu, click *Properties*.

The Motion Component dialog appears.



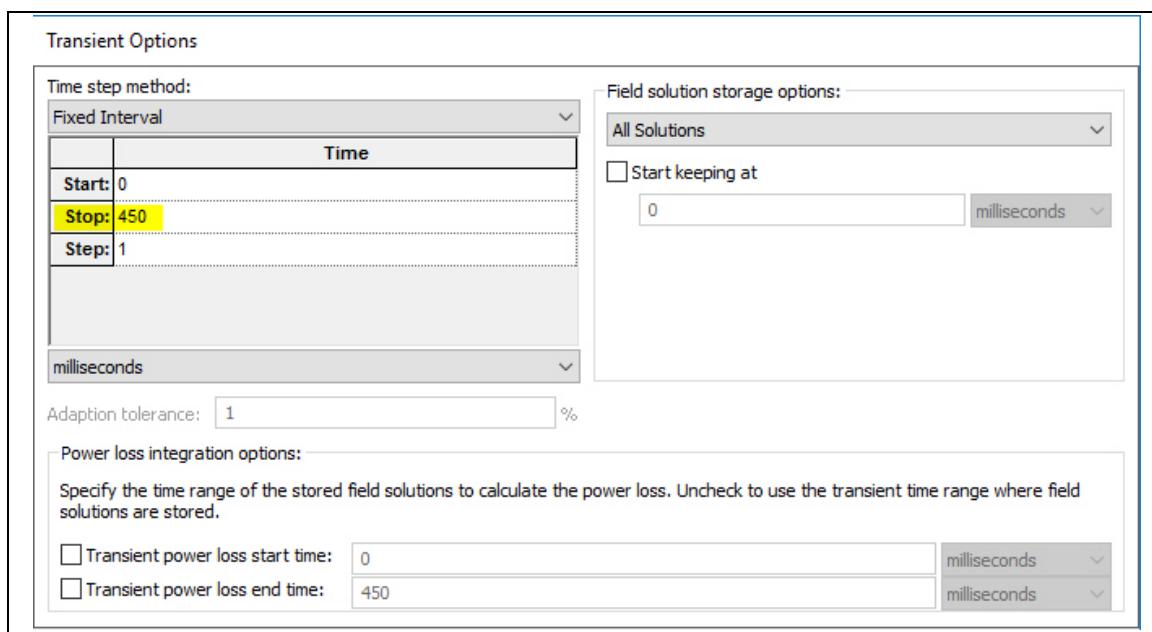
3. In the General page, set the *Source Type* as **Load driven**.
4. Click OK.

11.2 Modifying the Transient options

In this procedure, we will change the *Stop* time to be *450 ms.*; this will allow the rotor to reach synchronous speed.

1. On the Solve menu, click *Set Transient Options*.

The Transient Options dialog appears.



2. Make sure that Fixed Interval is selected as the Time Step Method, and then make the following modification for Stop Time:
 - Start = 0 Milliseconds
 - Stop = 450 Milliseconds
 - Step = 1 Milliseconds
3. Click OK.

11.3 Solve the model

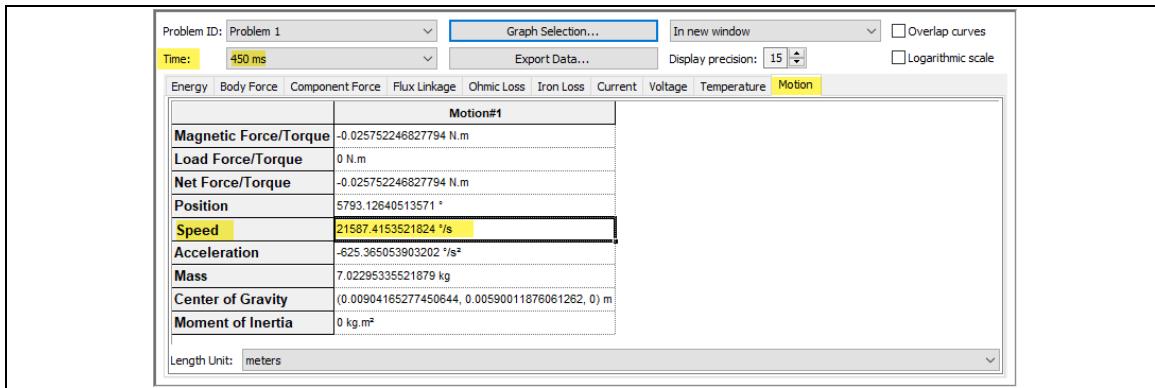
- On the Solve menu, click *Transient 2D with Motion*.

The Transient 2D with Motion Solver Progress dialog appears.

12 Analyzing the results for rotor speed and torque during start-up

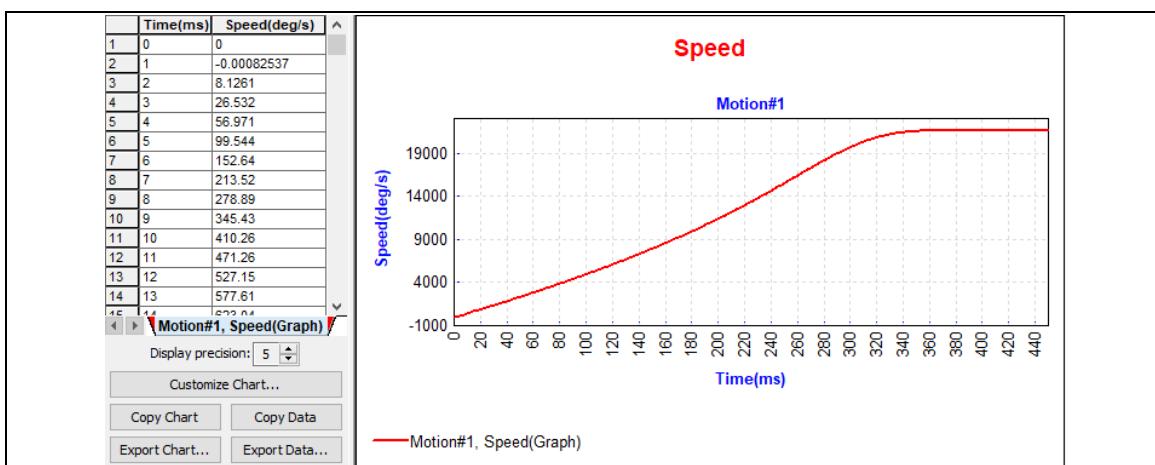
12.1 Graph the speed and magnetic torque

The Results window is automatically displayed when the solution is complete. Once the solution process is complete, it is possible to examine the speed and magnetic torque of the rotor over the entire time, from standstill to synchronous speed.



1. In the Results window, open the Motion page.
2. Select *450 ms* from the *Time* drop-down list.
3. Using the mouse pointer, click anywhere inside the *Speed* text box (21587.4153521824 deg/s).
Note This is equivalent to 377 rad/s, which is expected from this device.
4. Click the *Graph Selection* button, located at the top of the Results window.

This graph shows the speed of the rotor starting from zero, at time zero, and then increasing until synchronous speed is reached. It should look similar to the illustration below.



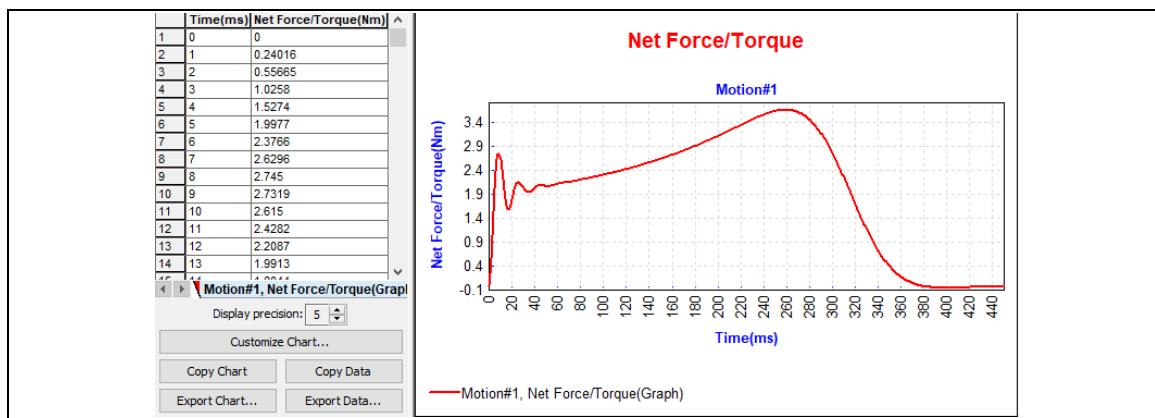
5. Switch back to the Results window by clicking the Results tab located at the bottom of the window.

6. Using the mouse pointer, click anywhere inside the Net Force/Torque text box (i.e., -0.0257528800750344 N.m).

Note At 450 ms, the torque should be zero. The value (-0.0257528800750344 N.m) is an indication of numerical and round-off errors.

7. Click the *Graph Selection* button.

This graph displays the torque that the rotor experiences going through some initial transient and then reaching a maximum before going to zero at synchronous speed. It should look similar to the illustration below.



8. On the File menu, click *Save*.
9. On the File menu, click *Close*.

13 Summary

In this tutorial, you completed the steps in creating a half model for time-harmonic and transient solutions. The skills you learned include:

- Setting up the work environment by modifying initial settings and the viewing area.
- Building the geometric model using the Keyboard Input Bar.
- Transforming construction slice edges.
- Creating new user materials.
- Setting up the problem, which consists of making components and
- Creating coils, assigning boundary conditions, modifying the mesh, and creating circuits.
- Making a motion component.
- Generating the time-harmonic and transient field solutions using
- Simcenter MAGNET's Time-Harmonic 2D and Transient 2D with Motion solvers.
- Analyzing the results, which includes:
 - viewing the torque of the locked rotor and viewing its contour plot
 - graphing the torque for the rotor when it is rotating at 200 rad/s
 - graphing the speed and torque of the rotor during the start-up phase

13.1.1 References

¹Kent Davey, *Induction Motor Analyses*, International TEAM Workshop Problem 30

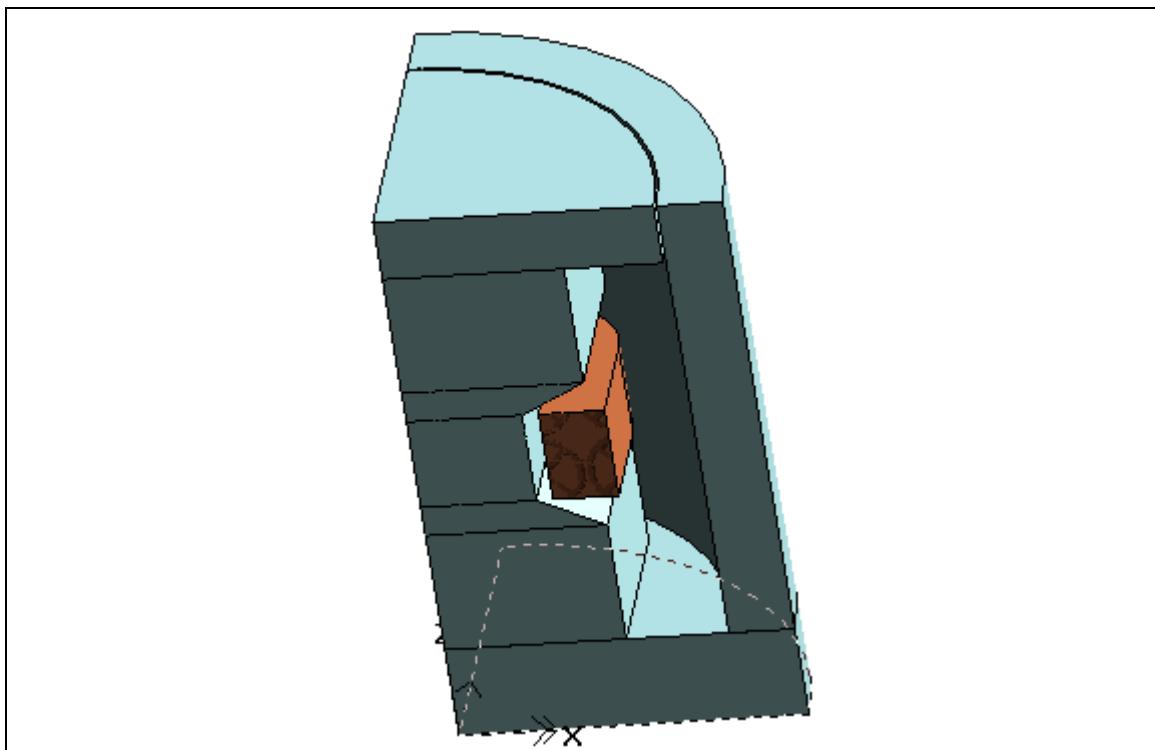
Tutorial #7

3D Magnetostatic
Pot-core with a coil

1 Modeling plan

The pot-core is symmetrical about the X and Y-axes and also about the 45 degree X-Y diagonal. Although these symmetry conditions allow for only one-eighth of the model to be built, this tutorial models one-quarter of the pot-core (one-quarter of the model is easier to visualize).

The pot-core is built from eight components. There is one additional component for the coil. One final component forms an air box that surrounds the model. (The air box is not shown in the diagram below.)



2 Open a new model

- Start Simcenter MAGNET.

If Simcenter MAGNET is already running, select New from the File menu to open a new model.

2.1 Name the model

1. On the File menu, click *Save As*.
2. In the *Save As* dialog box, enter **Pot-core with coil**.
3. Choose the drive and directory where you want to place the model.
4. Click *Save As*.

2.2 Set the model units

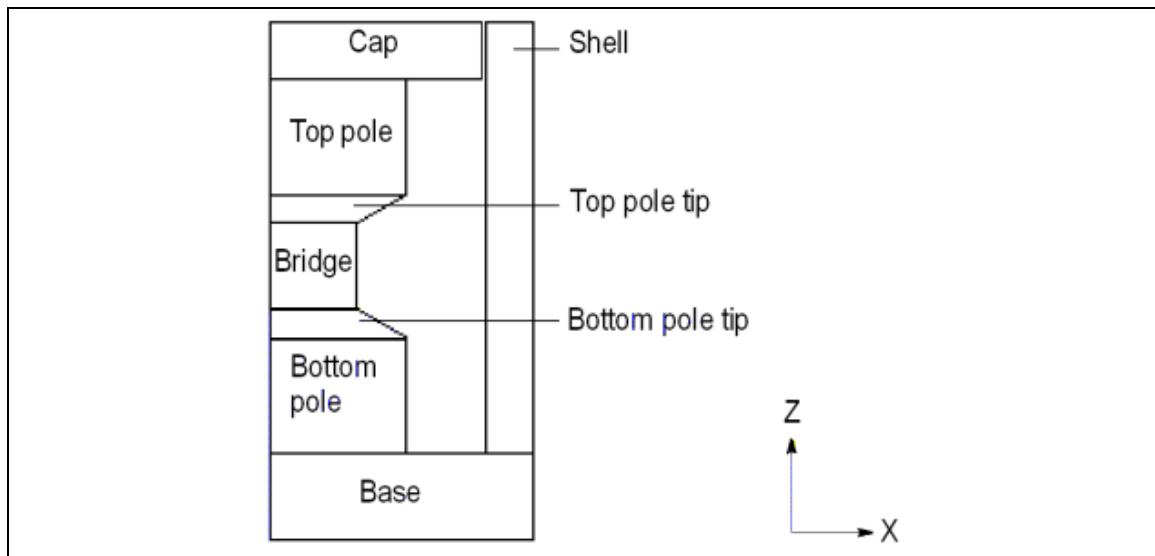
The Simcenter MAGNET default unit of measurement is meters. The pot-core will be modeled in centimeters.

You can set centimeters to be preferred unit of measurement in all the Simcenter MAGNET dialogs. This option is set in the General Model property page.

1. In the Object page of the Project bar, select the model (i.e. *Pot-core with coil.mn*).
2. On the Edit menu, click *Properties*.
The Properties dialog appears.
3. On the Units page, in the *Length* drop down list, select **Centimeters**.
4. Click OK.

3 Build the geometric model

The pot-core is built from eight components.



4 Create the base of the pot-core

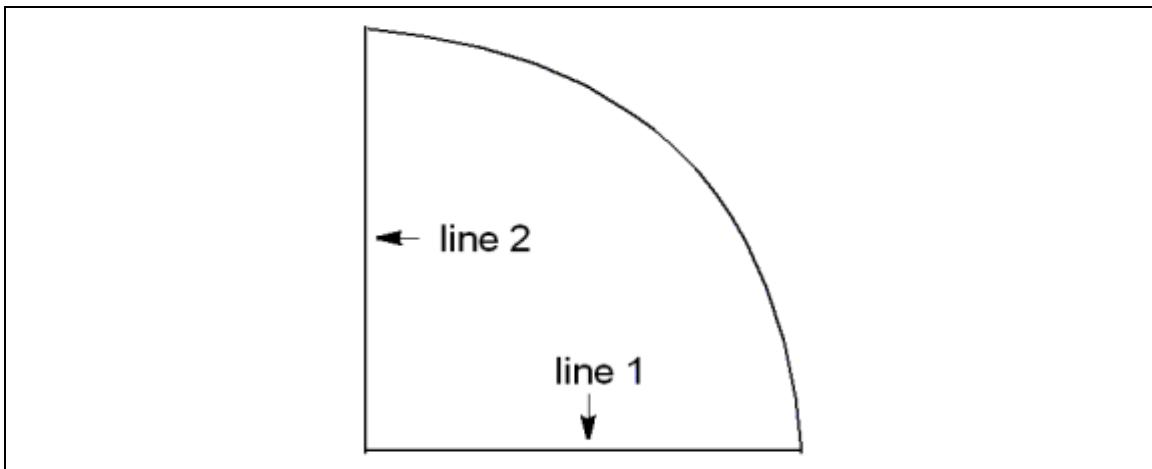
The first component forms the base of the pot-core.

4.1 Draw the geometry of the base

Drawing tools are located on the Draw toolbar and the Draw menu. Edges are drawn by using the mouse pointer or by entering coordinates through the Keyboard Input bar. In this tutorial, edges are drawn using the Keyboard Input bar.



The geometry for the base is shown in the diagram below.



1. If the Keyboard Input bar is not already displayed at the bottom of the Main window, select Keyboard Input Bar on the Tools menu.

2. Make sure that (Cartesian) and (Absolute) are selected on the Keyboard Input bar.
3. On the Draw toolbar, click (Line drawing tool).
4. On the View menu, click *Update Automatically*.
5. In the Keyboard Input bar, enter the following coordinates for line 1.

Start coordinates 0, 0 Press ENTER

End coordinates 10.6, 0 Press ENTER

6. Press ESC.

7. In the Keyboard Input bar, enter the following coordinates for line 2.

Start coordinates 0, 0 Press ENTER

End coordinates 0, 10.6 Press ENTER

8. Press ESC.

9. On the Draw toolbar, click .

10. In the Keyboard Input bar, enter the following coordinates.

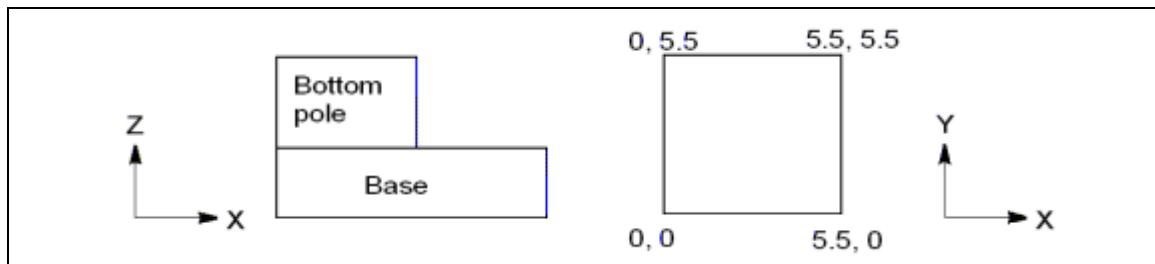
Center coordinates	0, 0	Press ENTER
Start coordinates	10.6, 0	Press ENTER
End coordinates	0, 10.6	Press ENTER

4.2 Make the component

1. On the Selection toolbar, click  (Select Construction Slice Surfaces tool).
2. Click the mouse pointer inside the surface of the base.
3. On the Model toolbar, click  (Make Component in a Line tool).
4. In the *Name* box, enter **Base**.
5. In the *Material* drop down list, select **MU3: Relative permeability 1000**.
6. In the *Distance* box, enter **3**.
7. Click **OK** to accept the settings.
8. Click **Save**.

5 Create the bottom pole

The next component is the base of the first pole of the pot-core.



The bottom pole begins at the ending surface of the base. The construction slice is moved to the ending surface of the Base before the geometry is drawn.

5.1 Move the construction slice

1. On the Object page, select *Base, Face#2 (End Face)*.
2. On the Draw toolbar, click  (Move Construction Slice tool).
3. In the Move Construction Slice dialog box, make sure that **To The Currently Selected Surface** is selected.
4. Click **OK**.

5.2 Draw the geometry

1. On the Draw toolbar, click  (Line Drawing tool).
2. In the Keyboard Input bar, enter the following coordinates for the polyline.

Start coordinates	0, 0	Press ENTER
End coordinates	5.5, 0	Press ENTER
End coordinates	5.5, 5.5	Press ENTER
End coordinates	0, 5.5	Press ENTER
End coordinates	0, 0	Press ENTER

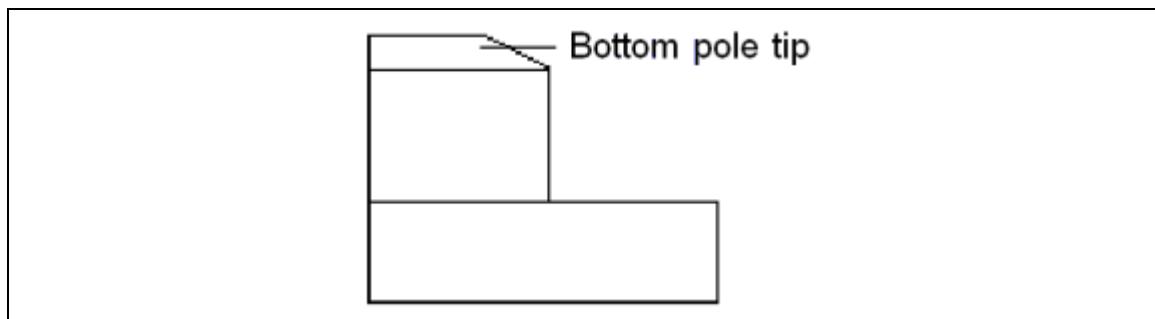
3. Press ESC.

5.3 Make the component

1. On the Selection toolbar, click  (Select Construction Slice Surfaces tool).
2. Click the mouse pointer inside the *construction slice surface* of the bottom pole.
3. On the Model toolbar, click  (Make Component in a Line tool).
4. In the *Name* box, enter **Bottom pole**.
5. In the *Material* drop down list, select **MU3: Relative permeability 1000**.
6. In the *Distance* box, enter **4**.
7. Click OK.

6 Create the bottom pole tip

The bottom pole tip is the tapered tip of the bottom pole. The tip begins at the ending surface of the bottom pole.



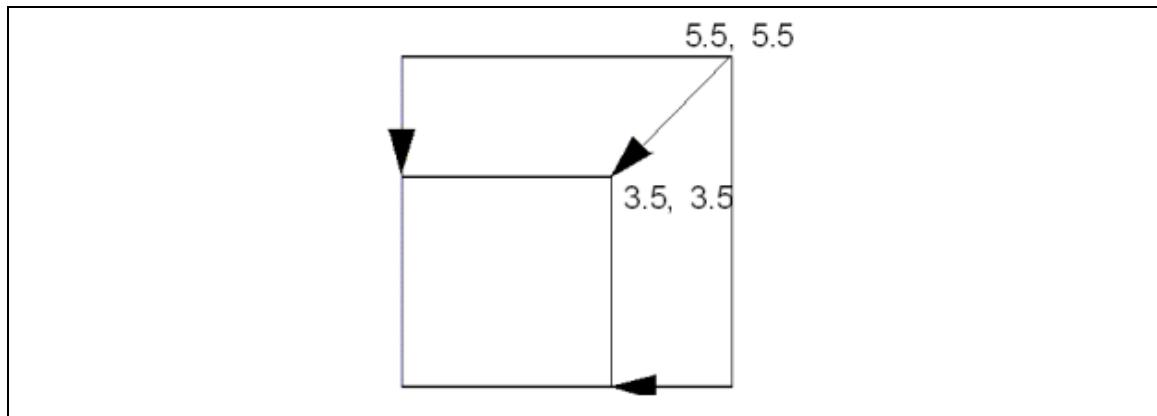
The pole tip is created in two steps. In the first step, a component is created from the surface of the bottom pole. In the second step, the geometry of the new component is distorted.

6.1 Make the component

1. On the Object page, select Bottom pole, Face#2 (End Face).
2. On the Model toolbar, click  (Make Component in a Line tool).
3. In the **Name** box, enter **Bottom pole tip**.
4. In the *Material* drop down list, select **MU3: Relative permeability 1000**.
5. In the *Distance* box, enter **1**.
6. Click **OK**.

6.2 Distort the shape of the bottom pole tip

The tip of the pole tapers inward from its starting surface to its ending surface. To create this effect, the vertices of the ending surface are moved.



6.3 Move the construction slice

1. On the Object page, select Bottom pole tip, Face#2 (End face).
2. On the Draw toolbar, click  (Move Construction Slice tool).
3. In the Move Construction Slice dialog box, make sure that **To The Currently Selected Surface** is selected.
4. Click **OK**.

6.4 Distort vertex (5.5, 5.5)

1. On the Model menu, click *Distort Vertices*.
The vertices of the model are displayed.
2. In the Keyboard Input bar, type **5.5, 5.5** (the original coordinate of the selected vertex) and press ENTER.
The vertex is highlighted.
3. In the Keyboard Input bar, type **3.5, 3.5** (the new coordinate for the selected vertex) and press ENTER.
The geometry of the model is updated.

6.5 Distort vertex (0, 5.5)

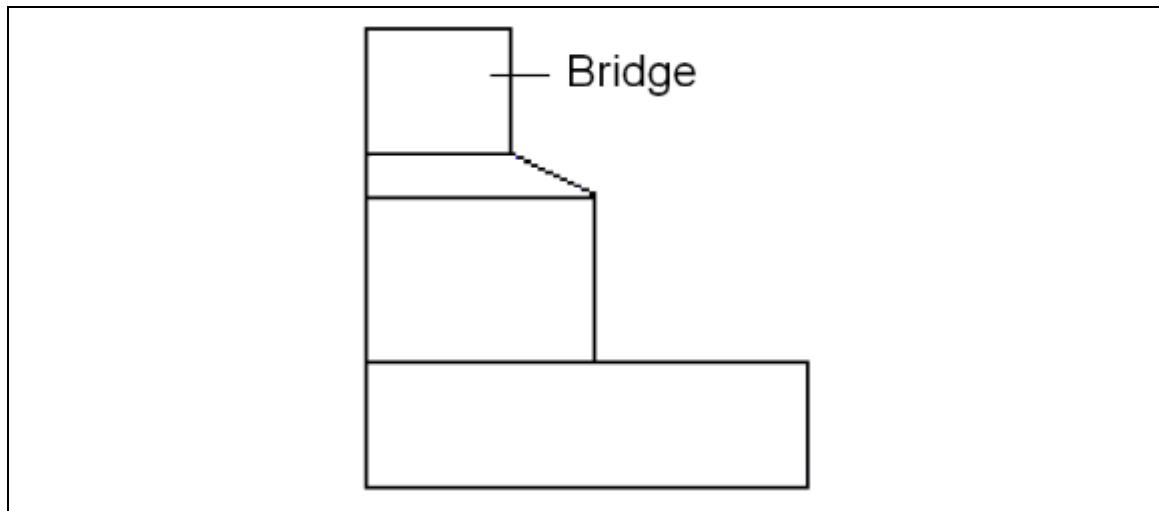
1. In the Keyboard Input bar, type **0, 5.5** (the original coordinate of the selected vertex) and press ENTER.
2. In the Keyboard Input bar, type **0, 3.5** (the new coordinate for the selected vertex) and press ENTER.

6.6 Distort vertex (5.5, 0)

1. In the Keyboard Input bar, type **5.5, 0** (the original coordinate of the selected vertex) and press ENTER.
2. In the Keyboard Input bar, type **3.5, 0** (the new coordinate for the selected vertex) and press ENTER.
3. On the File menu, click *Save*.

7 Create the bridge between the pole tips

A square block bridges the gap between the pole tips.

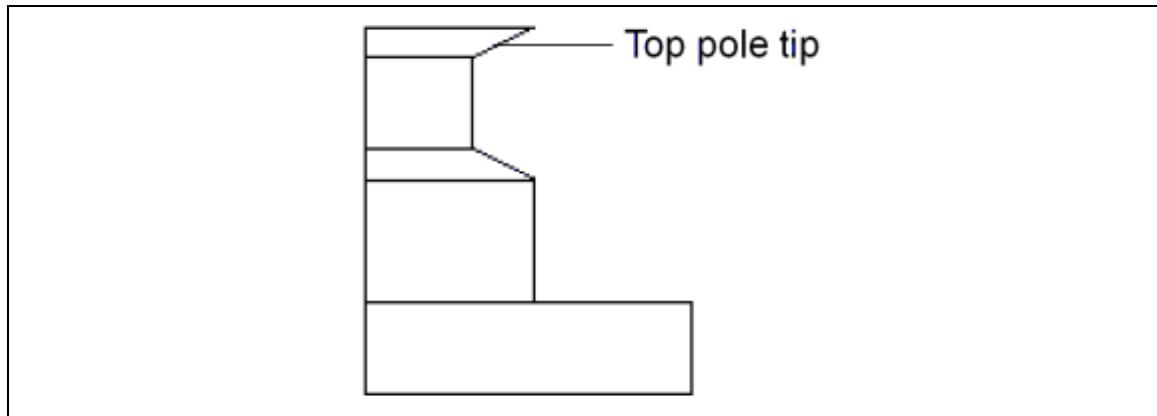


7.1 Make the component

1. On the Object page, select Bottom pole tip, Face#2 (End Face).
2. On the Model toolbar, click (Make Component in a Line tool).
3. In the *Name* box, enter **Bridge**.
4. In the *Material* drop down list, select **MU3: Relative permeability 1000**.
5. In the *Distance* box, enter **3**.
6. Click **OK**.

8 Create the top pole tip

The next component is the second pole tip.



8.1 Make the component

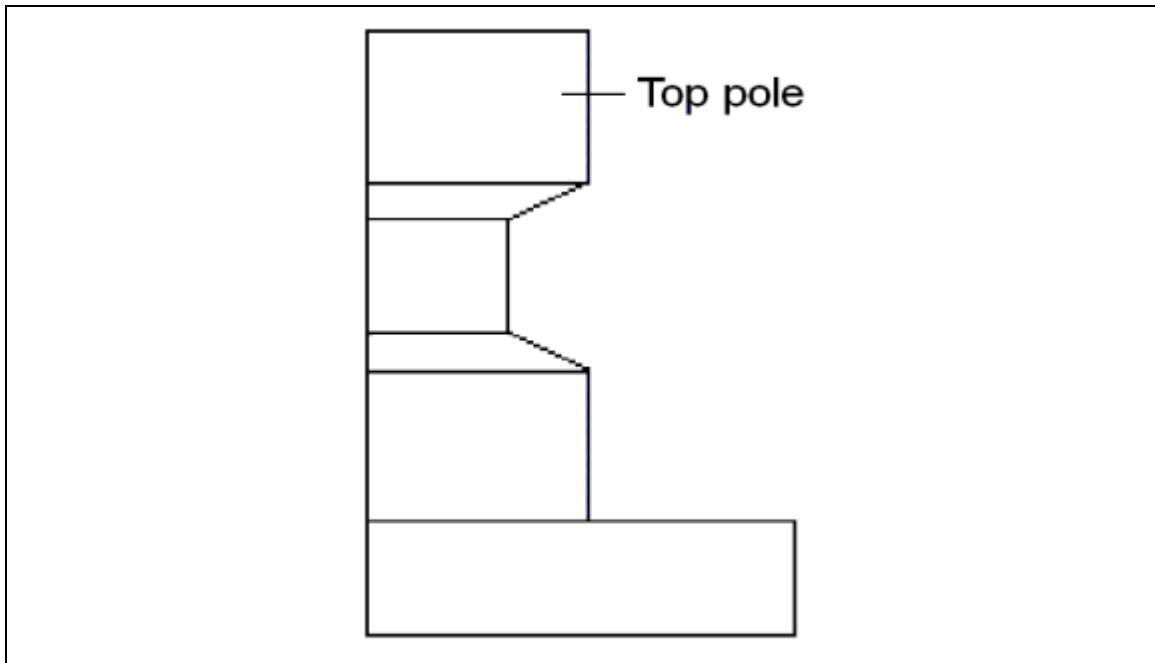
1. On the Object page, select *Bridge, Face#2 (End Face)*.
2. On the Model toolbar, click (Make Component in a Line tool).
3. In the *Name* box, enter **Top pole tip**.
4. In the *Material* drop down list, select **MU3: Relative permeability 1000**.
5. In the *Distance* box, enter **1**.
6. Click **OK**.

8.2 Distort the shape of the top pole tip

1. Move the construction slice to the end surface of the Top pole tip [i.e. Face#2 (End Face)].
2. On the Model menu, click *Distort Vertices*.
3. In the Keyboard Input bar, type **3.5, 3.5** (the original coordinate of the selected vertex) and press ENTER.
4. In the Keyboard Input bar, type **5.5, 5.5** (the new coordinate for the selected vertex) and press ENTER.
5. In the Keyboard Input bar, type **0, 3.5** and press ENTER.
6. In the Keyboard Input bar, type **0, 5.5** as the new coordinate and press ENTER.
7. In the Keyboard Input bar, type **3.5, 0** and press ENTER.
8. In the Keyboard Input bar, type **5.5, 0** as the new coordinate and press ENTER.
9. Select *Save*.
10. On the Edit menu, click *Select*.
The vertices are no longer displayed.

9 Create the top pole

The top pole begins at the ending surface of the top pole tip.



9.1 Make the component

1. On the Object page, select Top pole tip, Face#2 (End Face).
2. On the Model toolbar, click (Make Component in a Line tool).
3. In the *Name* box, enter **Top pole**.
4. In the *Material* drop down list, select **MU3: Relative permeability 1000**.
5. In the *Distance* box, enter **4**.
6. Click **OK**.

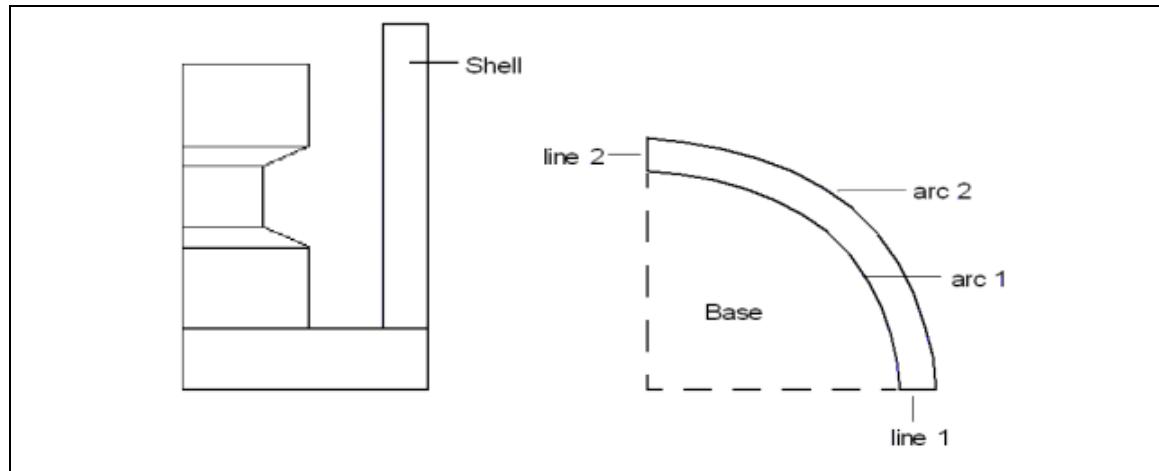
9.2 Delete Construction Slice edges

Since the edges on the construction slice remain even after being swept into a component, we must remove them before proceeding to the next step of the tutorial.

1. On the Edit menu, click Select Construction Slice Edges.
2. On the Edit menu, click *Select All*.
3. On the *Standard Tools* toolbar, click (Delete).

10 Create the shell

The shell begins at the ending surface of the base.



10.1 Draw the geometry

1. Move the construction slice to the end face of the Base [i.e. Face#2 (End Face)].
2. On the Draw menu, click *Line*.
3. On the View menu, click *Update Automatically*.
4. Draw the geometry using the following coordinates.

Line 1:

Start coordinates 10.6, 0
End coordinates 8.6, 0
Press ESC

Line 2:

Start coordinates 0, 10.6
End coordinates 0, 8.6
Press ESC

5. On the Draw menu, click *Arc (Center, Start, End)*.
6. Draw the geometry using the following coordinates.

Arc 1:

Center coordinates 0, 0
Start coordinates 8.6, 0
End coordinates 0, 8.6

Arc 2:

Center coordinates 0, 0
Start coordinates 10.6, 0
End coordinates 0, 10.6

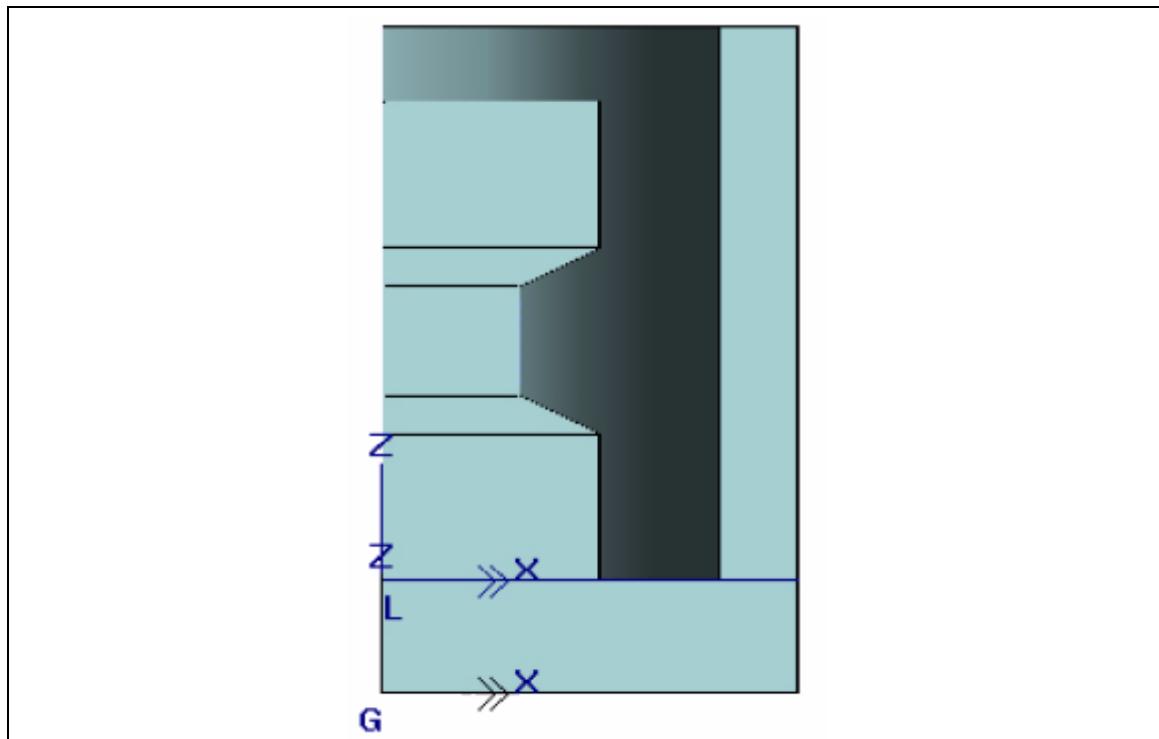
10.2 Make the component

1. On the Selection toolbar, click  (Select Construction Slice Surfaces tool).
2. Click the mouse pointer inside the *construction slice surface of the shell* (the area between the two arcs).
3. On the Model toolbar, click  (Make Component in a Line tool).
4. In the *Name* box, enter **Shell**.
5. In the *Material* drop down list, select **MU3: Relative permeability 1000**.
6. In the *Distance* box, enter **15**.
7. Click OK.

10.3 Rotate the model

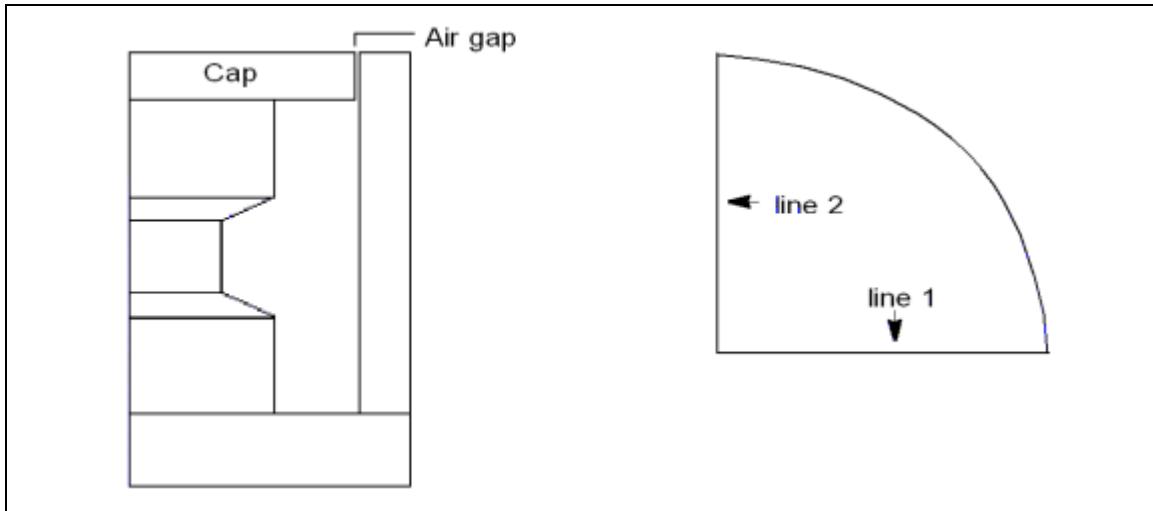
- On the Preset View toolbar, click  [Show XZ (-Y)].

The model should look like the diagram below.



11 Create the cap

The ending surface of the cap is located at the same sweep path position as the ending surface of the shell. The geometry of the cap can be drawn and then swept backwards (in the negative Z direction). Notice that there is a slight air gap between the cap and the shell.



11.1 Draw the geometry

1. Move the construction slice to the end surface of Shell [i.e. Face#2 (End Face)].
2. On the Preset View toolbar, click [Show XY (+Z)].
3. On the Draw menu, click *Line*.
4. Draw the geometry using the following coordinates.

Line 1:

Start coordinates 0, 0
End coordinates 8.5, 0
Press

Line 2:

Start coordinates 0, 0
End coordinates 0, 8.5
Press

5. On the Draw menu, click *Arc* (Center, Start, End).
6. Draw the geometry using the following coordinates.

Arc:

Center 0, 0
Start coordinates 8.5, 0
End coordinates 0, 8.5

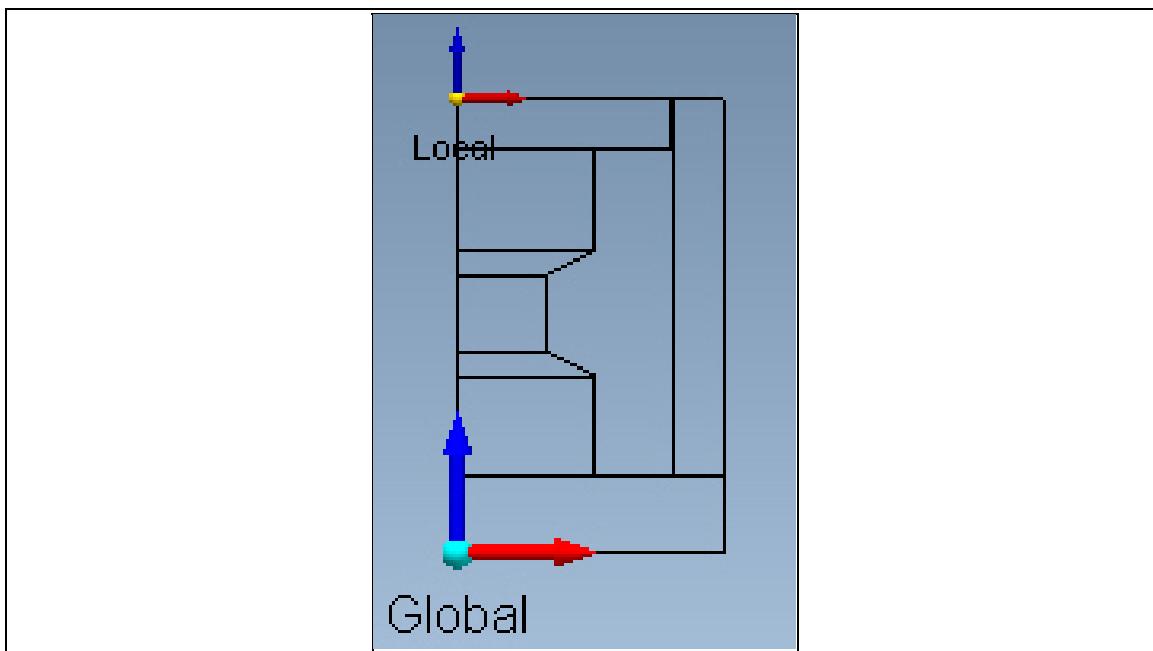
11.2 Make the component

1. On the Selection toolbar, click  (Select Construction Slice Surfaces tool).
2. Click the mouse pointer inside the *construction slice surface of the cap*.
3. On the Model toolbar, click  (Make Component in a Line tool).
4. In the *Name* box, enter **Cap**.
5. In the *Material* drop down list, select **MU3: Relative permeability 1000**.
6. In the *Distance* box, enter **-2**.
7. Click **OK**.
8. From the File menu, click **Save**.

11.3 Rotate the model

- On the Preset View toolbar, click  [Show XZ (-Y)].

The model should look like the diagram below.



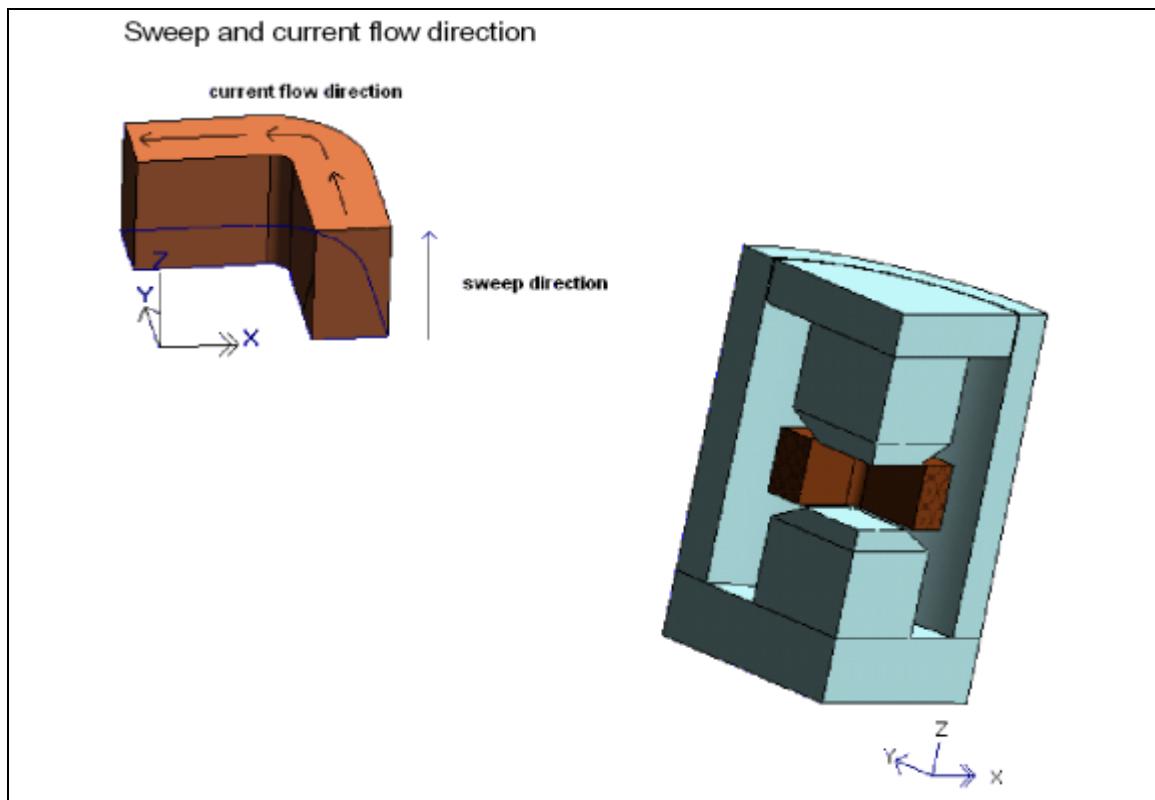
12 Create the excitation

In 3D, coils are created by selecting component surfaces (e.g. Face#1, Face#4) and then selecting the *Make Multi-Terminal Coil* command. The method chosen to make the coil depends on how the coil has been swept and how you want the current to flow in the coil.

Coil components can be swept in the direction of current flow, or swept in a direction normal to the current flow.

In this model, the coil is built from one component. The current flow follows the direction normal to the component sweep direction.

For more information on modeling *simple* and *multi-terminal* coils, please see the Help.



13 Create the component of the coil

13.1 Delete Construction Slice edges

Since the edges on the construction slice remain even after being swept into a component, we must remove them before proceeding to the next step of the tutorial.

1. On the Edit menu, click Select Construction Slice Edges.
2. On the Edit menu, click *Select All*.
3. On the *Standard Tools* toolbar, click  (Delete).

13.2 Hide the display of the shell

1. On the Object page, right-click the mouse pointer on *Shell*.
2. On the pop-up menu, toggle *Visible*.

13.3 Display a wireframe view

- On the View menu, click *Wireframe model*.

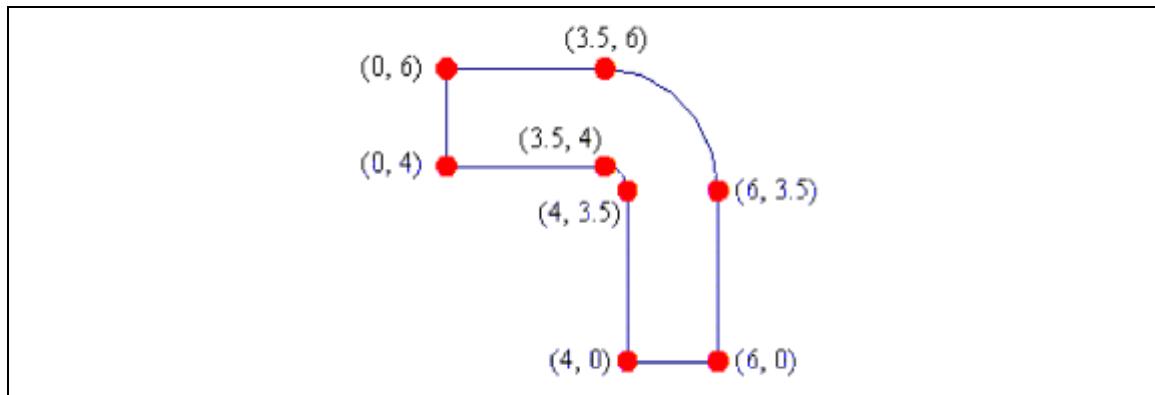
In a wireframe display, all the edges of a model are displayed without removing hidden surfaces.

13.4 Set the view

- On the View menu, click *Preset Views*, and then click *Positive Z Axis*.

13.5 Draw the geometry

The geometry of the coil component is shown below.



1. Move the construction slice to the start face (*Face#1*) of the Bridge.
2. On the Draw menu, click *Line*.

3. Draw the geometry using the following coordinates.

Line 1:

Start coordinates 4, 3.5

End coordinates 4, 0

End coordinates 6, 0

End coordinates 6, 3.5

Line 2:

Start coordinates 3.5, 4

End coordinates 0, 4

End coordinates 0, 6

End coordinates 3.5, 6

4. On the Draw menu, click *Arc (Center, Start, End)*.

5. Draw the geometry using the following coordinates.

Arc 1:

Center coordinates 3.5, 3.5

Start coordinates 4, 3.5

End coordinates 3.5, 4

Arc 2:

Center coordinates 3.5, 3.5

Start coordinates 6, 3.5

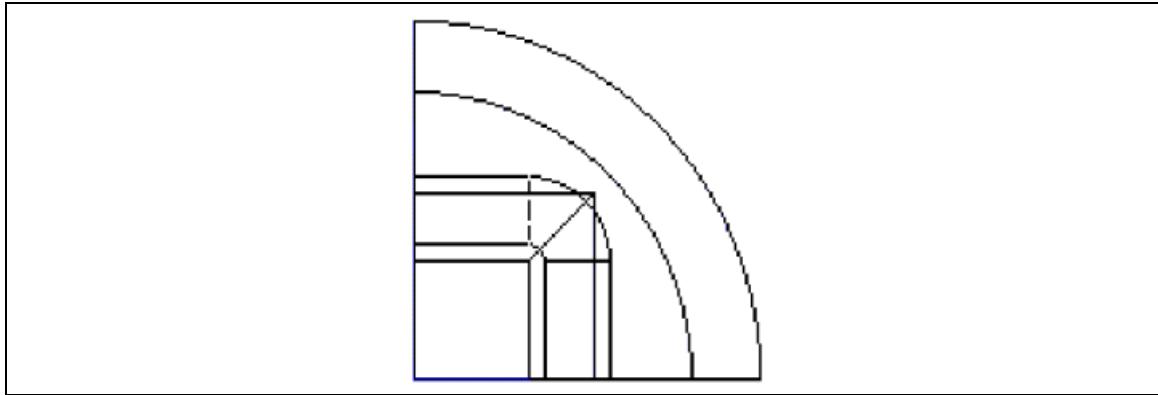
End coordinates 3.5, 6

13.6 Make the component

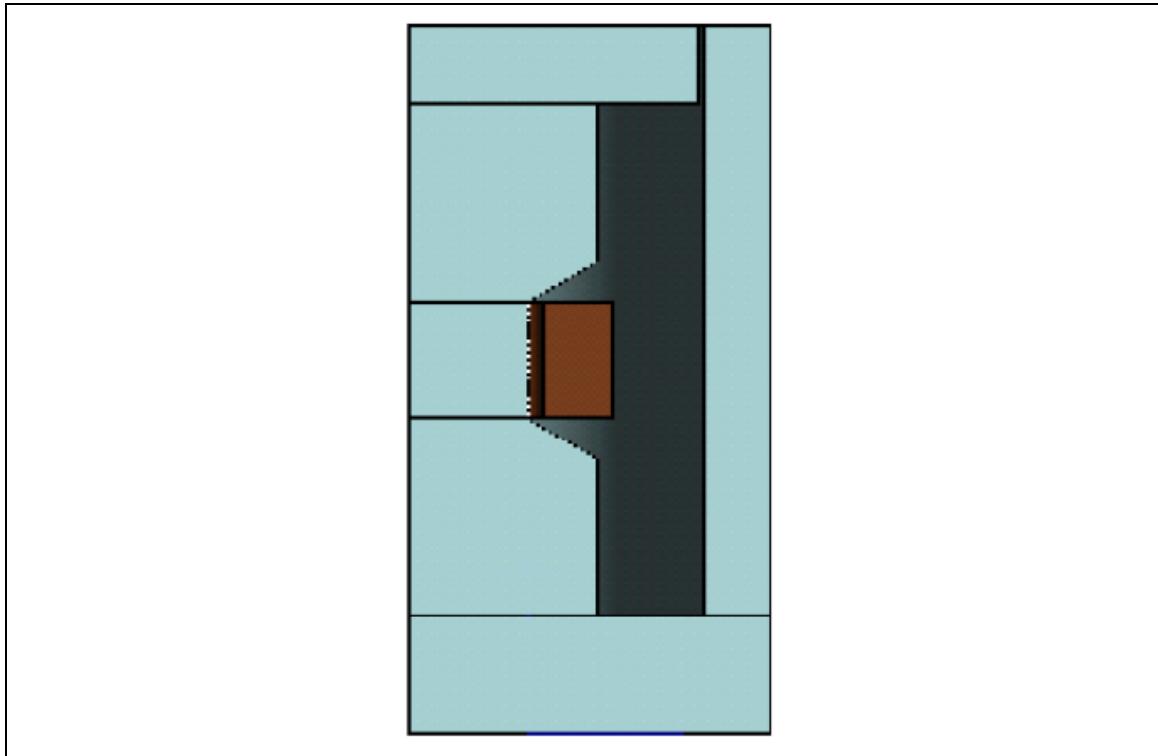
1. On the Edit menu, click Select Construction Slice Surfaces.
2. Click inside the geometric surface you have just created.
3. On the Model menu, click *Make Component in a Line*.
4. In the *Name* box, enter **Coil component**.
5. In the *Material* drop down list, select **Copper: 5.77e7 Siemens/meter**.
6. In the *Distance* box, enter **3**.
7. Click OK.

14 View the coil components

1. On the Preset View toolbar, click  [Show XY (+Z)].

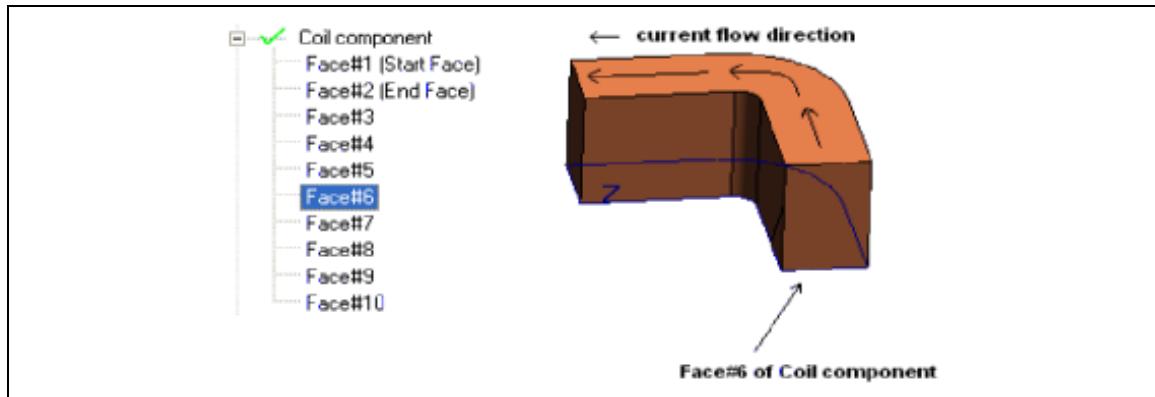


2. After verifying the shape of the coil components, re-display the Shell.
3. On the View menu, click *Solid Model*.
4. On the Preset View toolbar, click  [Show XZ (-Y)].



15 Create the coil

1. On the Object page, select *Face#6* and *Face #10* of the *Coil component*.



2. On the Model menu, click *Make Multi-Terminal Coil*.

The coil is listed in both the *Object page* and the *Coil page*.

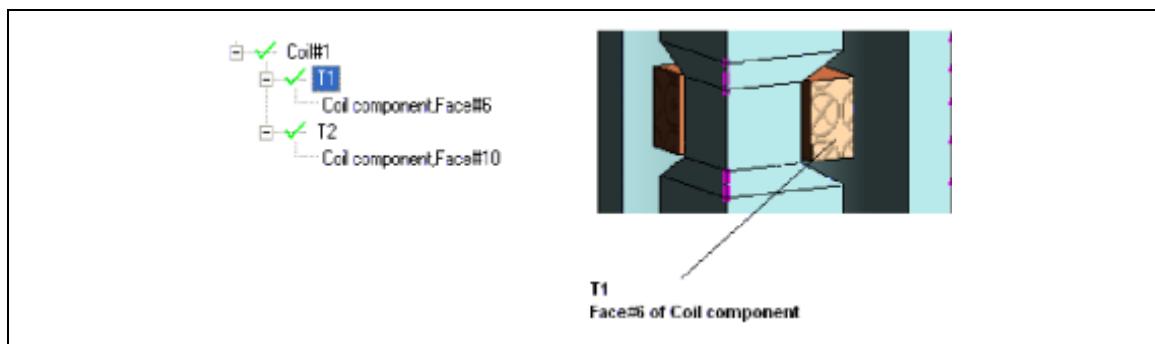
3. Click on the (+) sign of *Coil#1*.

The components of the coil are displayed.



4. Click on the (+) sign of *T1* and then select *T1*.

In the View window, the surface of *T1* is highlighted.

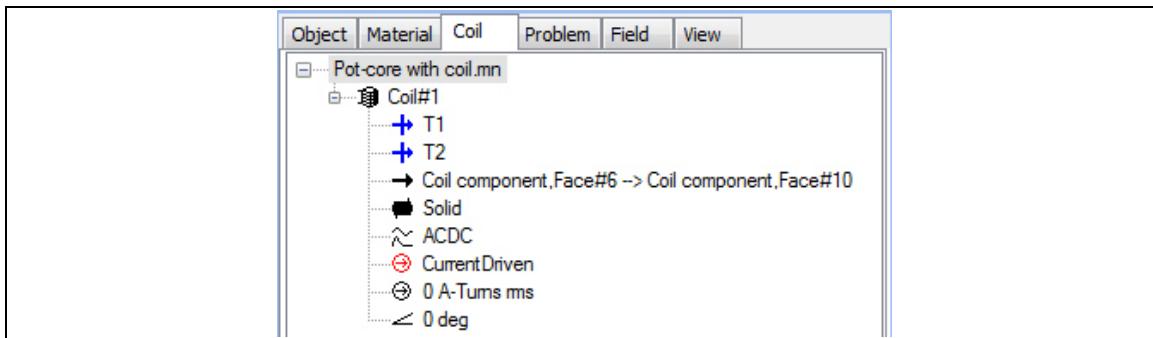


15.1 Change the current of the coil

In this problem, the coil should have 100 turns with 1 A per turn.

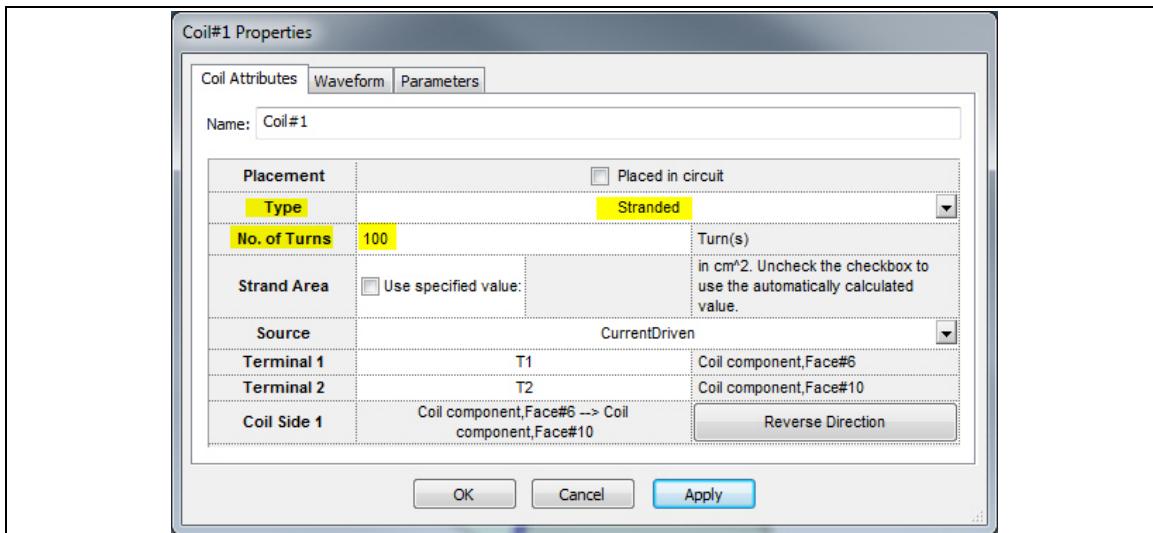
1. On the Project bar, select the *Coil* tab.

The *Coil* page appears.



2. On the Coil page, select *Coil#1*.
3. On the Edit menu, click *Properties*.

The *Coil Attributes* property page appears.



4. Do the following:
 - In the *Type* dropdown list, select **Stranded**.
 - In the *Number of Turns* text box, enter **100**.
5. Select the Waveform tab and enter the following values for DC:
Amplitude: **1**
6. Click OK.
The properties of the coil are updated on the Coil page.
7. Click Save.

15.2 Delete Construction Slice edges

Since the edges on the construction slice remain even after being swept into a component, we must remove them before proceeding to the next step of the tutorial.

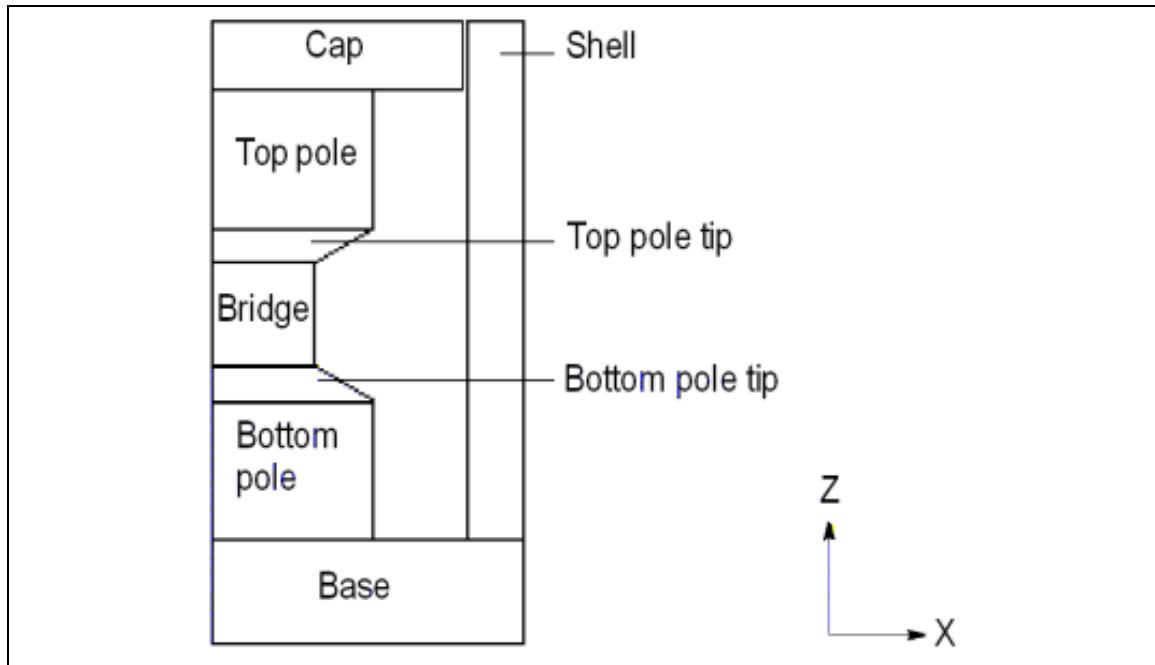
1. On the Edit menu, click Select Construction Slice Edges.
2. On the Edit menu, click *Select All*.
3. On the *Standard Tools* toolbar, click  (Delete).

16 Modify the mesh

In the 3D finite element method of analysis, the model is divided into a mesh of tetrahedral-shaped elements. Each element is defined by four vertices (nodes). The vector field inside each element is represented by a polynomial with unknown coefficients. The finite element analysis is the solution of the set of equations for the unknown coefficients.

The accuracy of the solution depends upon the nature of the field and the size of the mesh elements. In regions where the direction or magnitude of the field is changing rapidly, high accuracy requires small elements. One method of increasing mesh density is to set the *maximum element size* for a component volume or specific faces of a component. The following procedure will demonstrate this method.

A *maximum element size* modification of 1.1 centimeters will be applied to each component shown in the diagram below.

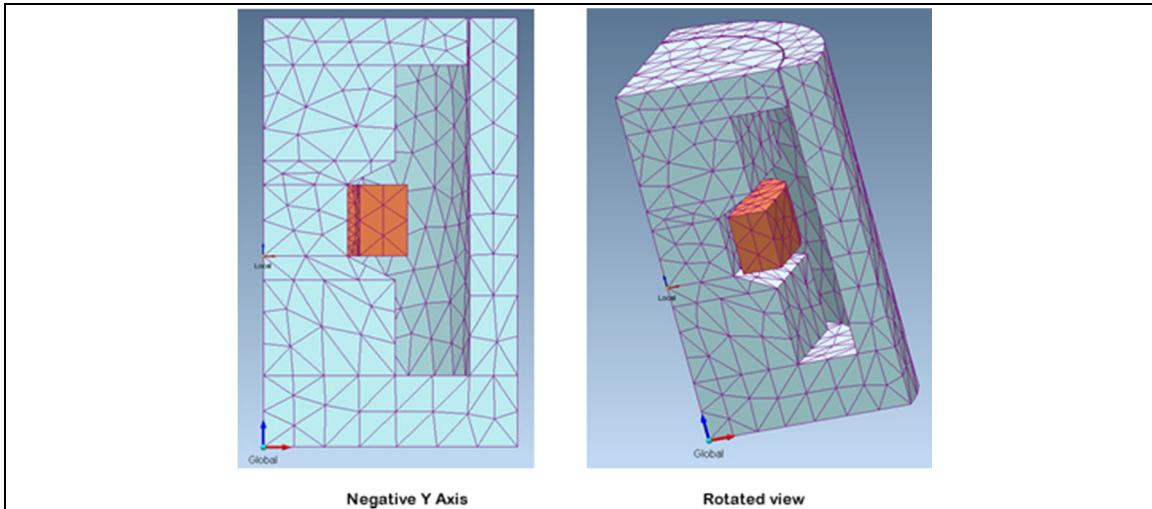


16.1 View the initial mesh

Before changing the *maximum element size*, the default initial mesh can be viewed.

1. On the Preset View toolbar, click  [Show XZ (-Y)].
2. On the View toolbar, click  (View All).
3. On the View menu, click *Initial 3D Mesh*.

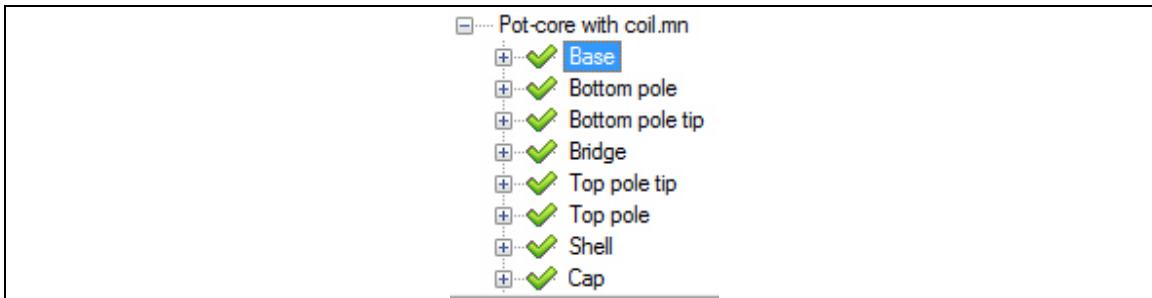
The initial mesh appears in the View window and should look like the following diagrams (shown here in both the “Negative Y axis” and “Rotated” views).



4. On the View menu, click *Solid Model*.

16.2 Set the maximum element size for each component

1. In the Object page of the Project bar, begin by selecting the *Base* component.

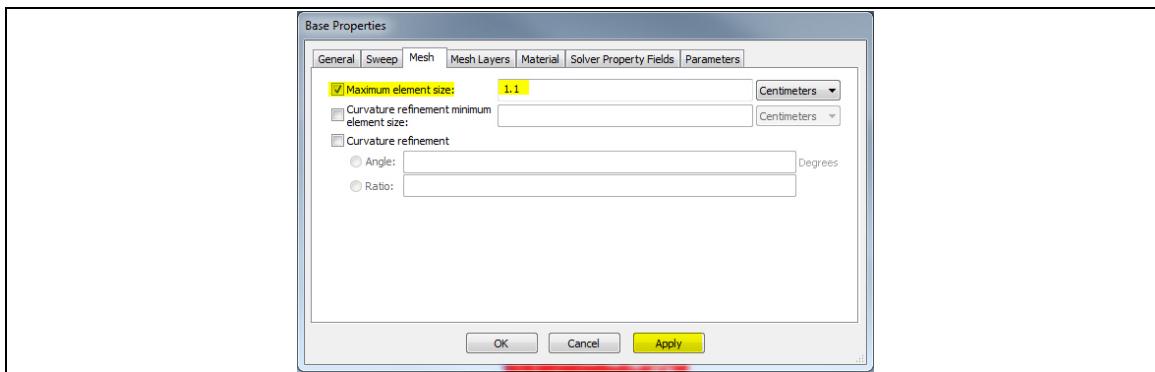


2. On the Edit menu, click *Properties*.

The Properties dialog appears.

3. Select the *Mesh* tab.

- Click inside the *Maximum element size* checkbox, and then type **1.1** in the text box.



- Click *Apply*.

Tip Clicking *Apply*, instead of *OK*, keeps the dialog open and allows us to proceed to the next component without having to repeat steps 2 and 3.

- In the Object page, select the *Bottom pole* component.

Notice that the text in the Properties dialog Title Bar has now changed to read *Bottom pole Properties*.

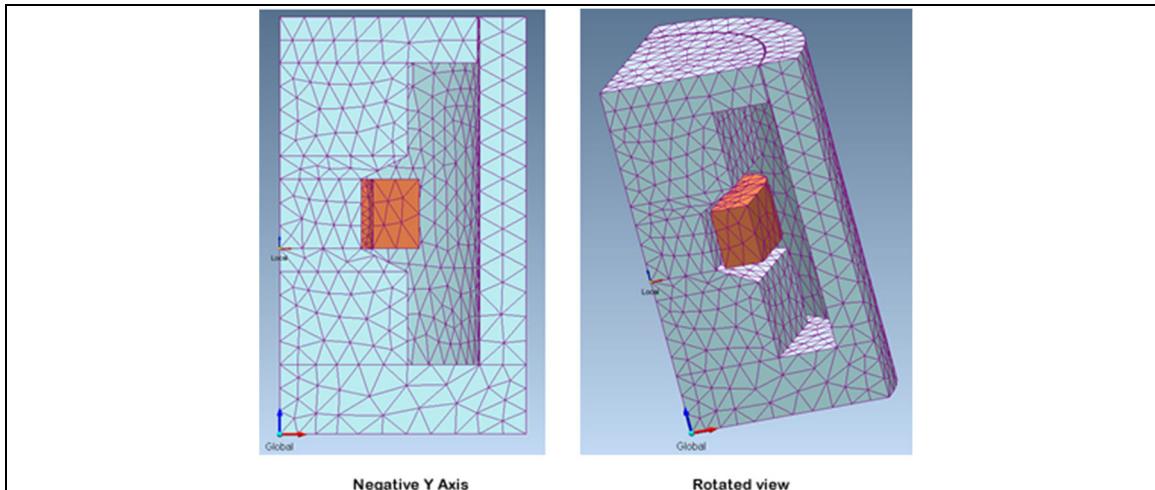
- Click inside the *Maximum element size* checkbox, and then type **1.1** in the text box.
- Click *Apply*.
- Repeat steps 6 through 8 for each of the remaining components (i.e. *Bottom pole tip*, *Bridge*, *Top pole tip*, *Top pole*, *Shell*, and *Cap*).

- Once all the *Maximum element size* modifications have been done, click *OK*.

16.3 View the changes to the mesh

- On the View menu, click *Initial 3D Mesh*.

The modified initial mesh appears in the View window and should look like the following diagrams (shown here in both the “Negative Y axis” and “Rotated” views).



- On the View menu, click *Solid Model*.

17 Add a boundary condition

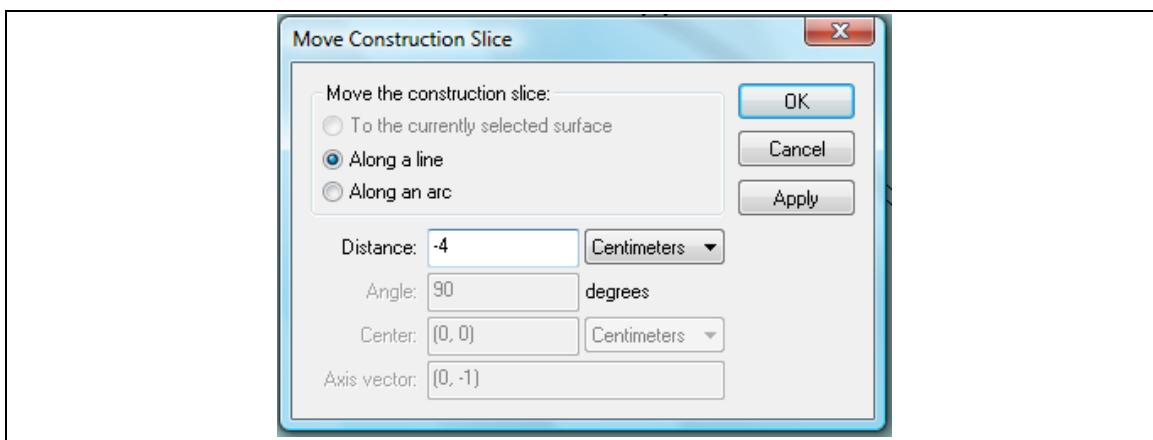
In this model, a tangential flux boundary is added around the pot-core. The boundary is created by sweeping an air box around the pot-core. The default boundary condition, tangential flux, is automatically applied to the outermost surfaces of the air box.

18 Create the air box

The starting surface of the air box begins 4 centimeters below the starting surface of the base.

18.1 Move the construction slice

1. From the Object page, select *Face#1 (Start face)* of the *Base* component.
2. On the Draw toolbar, click  (Move Construction Slice tool).
The Move Construction Slice dialog appears.



3. Verify that *To the currently selected surface* is selected.

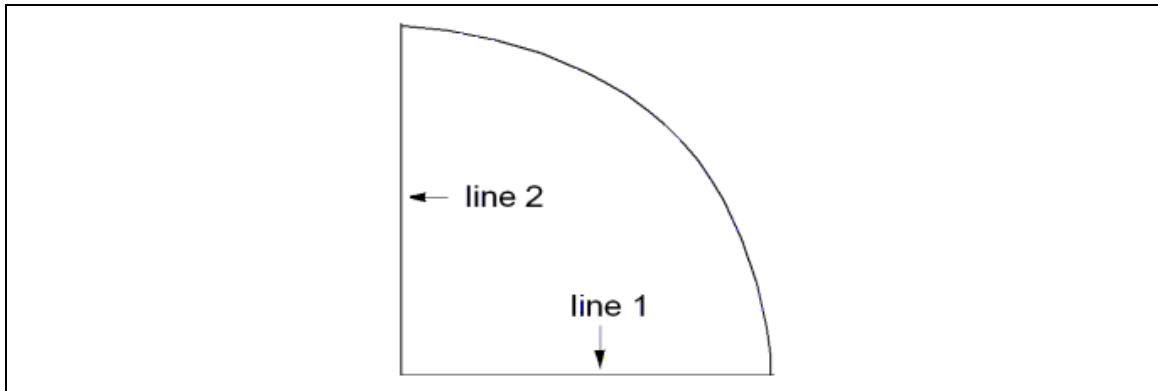
4. Click *Apply*.

Tip Clicking *Apply*, instead of *OK*, leaves the dialog open for the next step.

5. Select *Along A Line*.
6. In the *Distance* box, enter **-4**.
7. Click *OK*.

18.2 Draw the geometry

The air box is shaped like a quarter-circle surrounding the pot-core.



1. On the Preset View toolbar, click [Show XY (+Z)].
2. Draw the geometry using the following coordinates.

Line 1:

Start coordinates 0, 0
End coordinates 13.6, 0

Line 2:

Start coordinates 0, 0
End coordinates 0, 13.6
Arc (Center, Start, End):

Center coordinates 0, 0
Start coordinates 13.6, 0
End coordinates 0, 13.6

18.3 Make the component

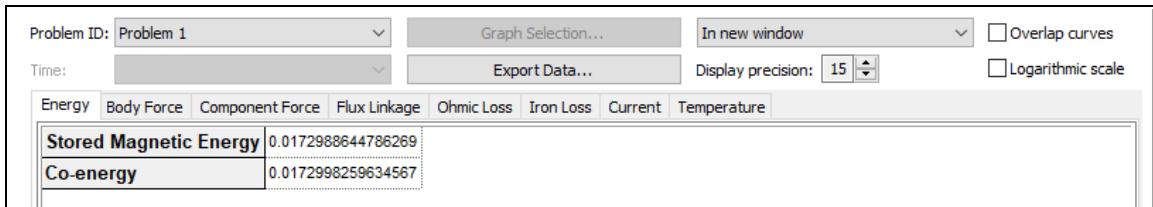
The air box extends 4 centimeters past the top of the pot-core.

1. On the Selection toolbar, click .
2. Click the mouse pointer inside the *surface of the air box*.
3. On the Model toolbar, click .
4. In the *Name* box, enter **Air box**.
5. In the *Material* drop down list, select **AIR**.
6. In the *Distance* box, enter **26**.
7. Click **OK**.
8. On the File menu, click **Save**.

19 Solve

- On the Solve menu, click *Static 3D*.

The *Static 3D Solver Progress* dialog appears briefly and then the Results window opens.



20 View the solution results

In this section, the following results are viewed:

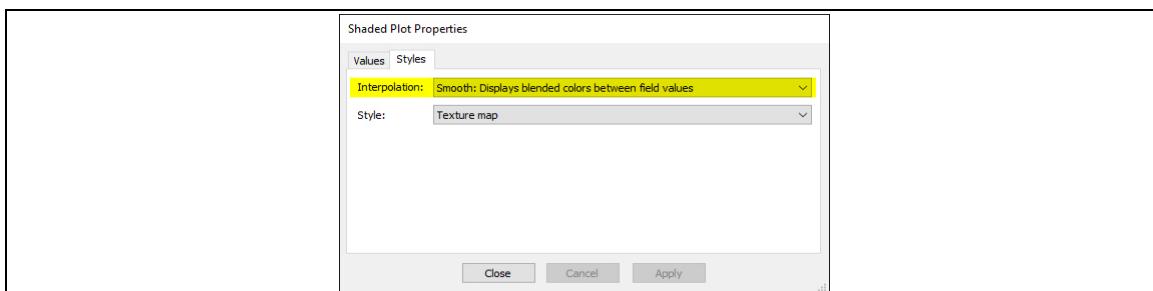
- Shaded plot of $|B|$ smoothed on the surface of the model and on a slice
- Arrow plot of B on the slice

20.1 Set the color interpolation and style of the shaded plot

This procedure will set the default for shaded plots to smooth instead of discrete, which is the default.

- On the View menu, click *Default Fields*.
- On the Project Bar, select the *View* tab.
- From the View tree, click *Shaded Plot*.
- On the Edit menu, click *Properties*.

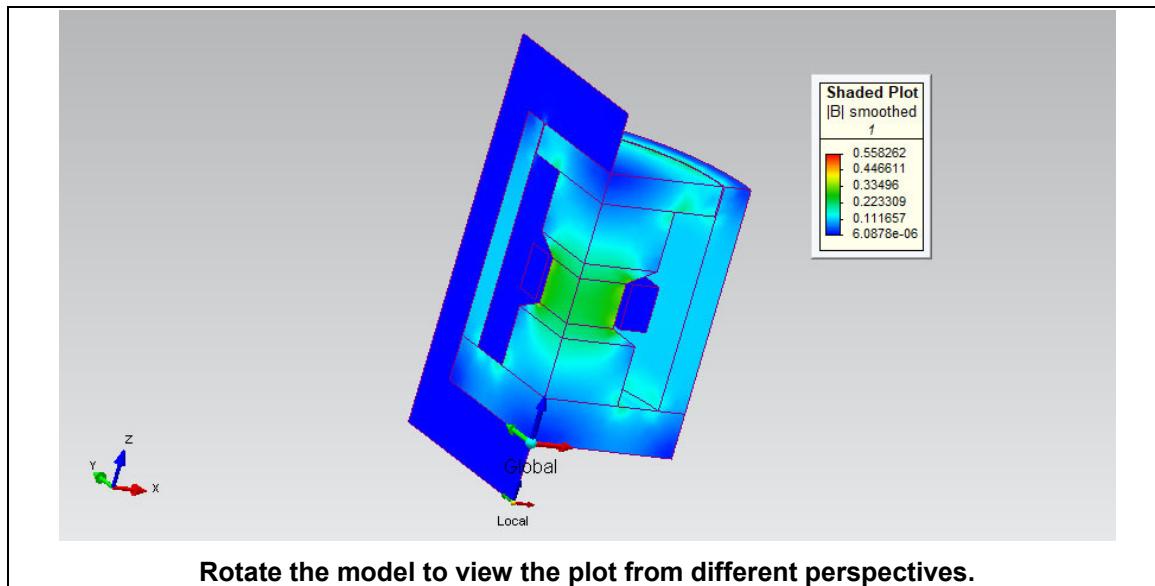
The Shaded Plot Properties page appears.



- Click the Styles tab and then from the Interpolation drop-down menu, select "Smooth: Displays blended colors between field values".
- Click OK.

20.2 View the shaded plot of $|B|$ smoothed

1. On the View menu, click *Preset Views*, and then click *Negative Y Axis*.
2. On the Object page, select *Air box*, right click the mouse, and then click *Visible*.
The  symbol next to the Air box component indicates that it is hidden.
3. Since $|B|$ smoothed is the default field for shaded plots, it was already displayed in the previous procedure (i.e. set the color interpolation and style of the shaded plot). The only difference will be that the Air box is hidden from the display.

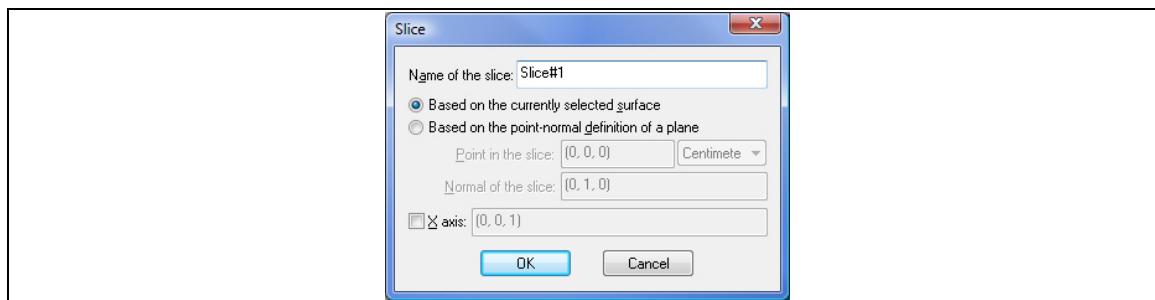


20.3 Create a slice

In this example, the slice will be created based on a surface of the model.

1. On the Object page, select *Base, Face#5*.
2. On the Tools menu, click *New Slice*.

The *Slice* dialog box appears.



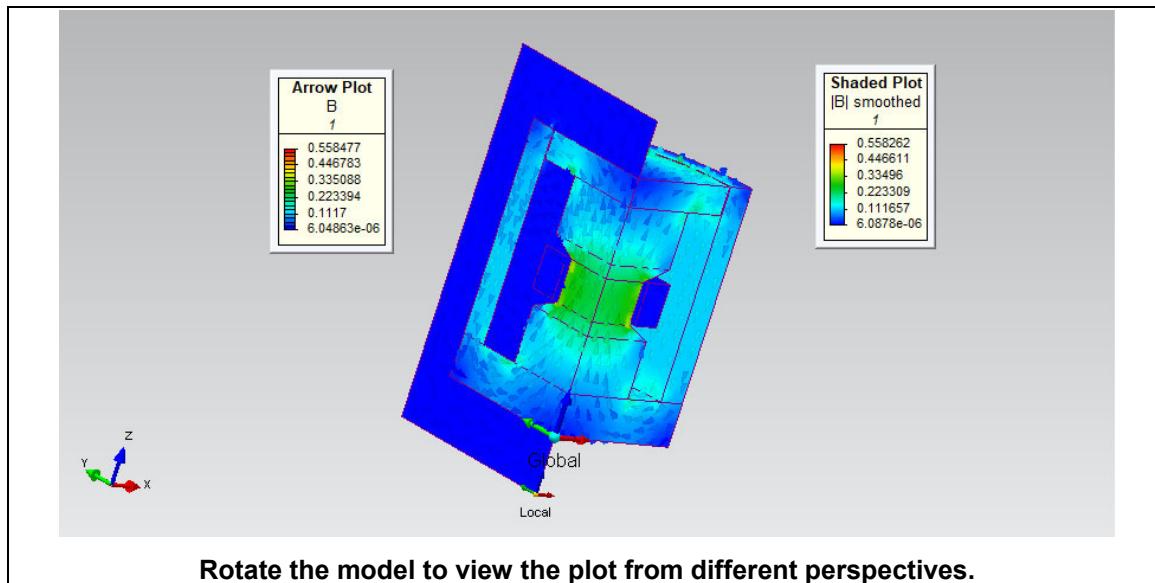
3. Make sure that Based on the currently selected surface is selected.
4. Select OK to create the slice.

Rotate the model. The shaded plot is automatically updated to display the slice.

20.4 View an arrow plot of B

1. On the *Field* page, select the *Arrow* tab at the bottom of the *Fields to Display* box.
2. In the *Fields to Display* box, select **|B| smoothed**.
3. Press Update View.

The arrow plot is displayed on the shaded plot and slice.



20.5 Save the model

You have now completed the pot-core tutorial.

1. On the File menu, click *Save*.
2. On the File menu, click *Close*.

21 Summary

In this tutorial, you built and solved a pot-core with a coil. The skills you learned include:

- Moving the construction slice
- Distorting the shape of a component
- Building a coil
- Refining the mesh by changing the *maximum element size*
- Viewing a shaded plot and an arrow plot

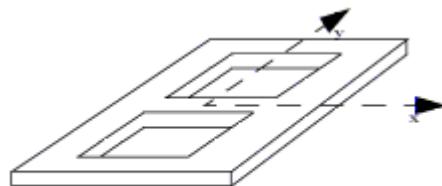
Tutorial #8

3D Time-harmonic

Bath plate

1 Modeling plan

The Bath plate is a conducting ladder with two holes symmetrically located at the center of the plate. A 50 Hz current-carrying coil is positioned above the plate. The time-harmonic solver computes the fields and induced eddy currents flowing in the conducting plate.

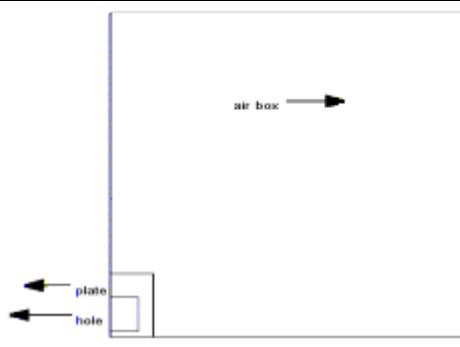


The Bath plate with symmetry planes

The plate is made of aluminum with a conductivity of $s = 0.3278 \times 10^8$. The exciting ampere-turns of the coil are 1260 at a frequency of 50 Hz.

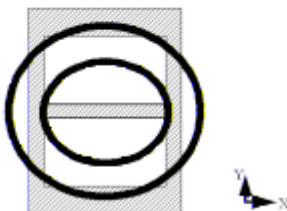
The model is symmetric about the X and Y planes, so only one-quarter of the model needs to be built and solved.

An air box is added that allows for the stray fields outside the Bath plate.



2 Coil

The primary coil above the Bath plate is a closely wound toroidal coil. The coil is located 15 millimeters above the Bath plate. The position of the coil in relation to the Bath plate is shown below.



The position of the primary coil (full model)

3 Open a new model

- Start Simcenter MAGNET.

If Simcenter MAGNET is already running, select New from the File menu to open a new model.

3.1 Name the model

1. On the File menu, click *Save As*.
2. In the *Save As* dialog box, enter **Bath plate**.
3. Choose the drive and directory where you want to place the model.
4. Click *Save As*.

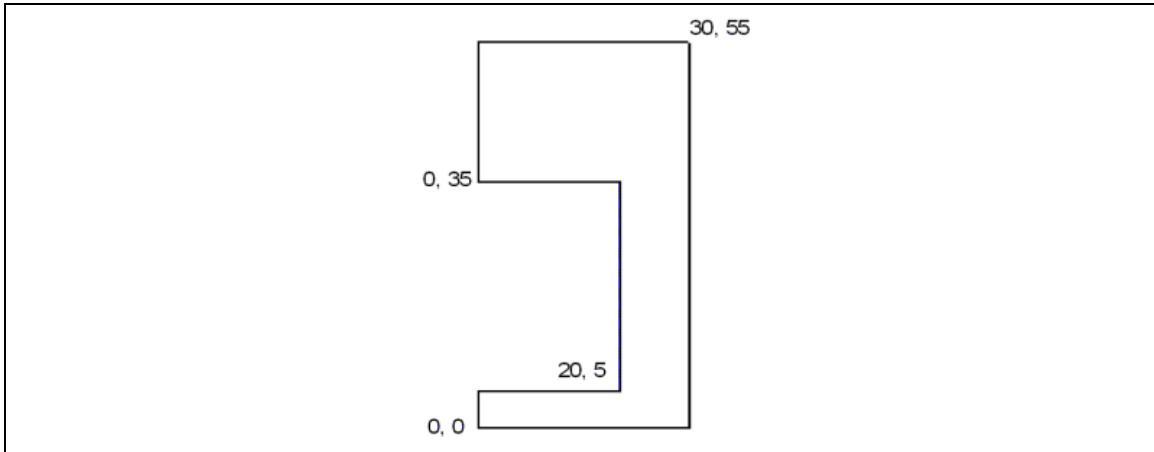
3.2 Set the model units

The Simcenter MAGNET default unit of measurement is meters. The Bath plate will be modeled in millimeters. You can set millimeters to be preferred unit of measurement in all the Simcenter MAGNET dialogs. This option is set in the General Model property page.

1. In the Object page of the Project bar, select the model (i.e. *Bath plate.mn*).
2. On the Edit menu, click *Properties*.
The Properties dialog appears.
3. On the Units property page, in the *Length* drop down list, select **Millimeters**.
4. Click *OK*.

4 Build the geometric model

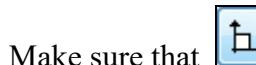
The Bath plate is built from one component.



4.1 Draw the geometry

Drawing tools are located on the Draw toolbar and the Draw menu. Edges are drawn by using the mouse pointer or by entering coordinates through the Keyboard Input bar.

1. If the Keyboard Input bar is not already displayed at the bottom of the Main window, select Keyboard Input Bar on the Tools menu.



Make sure that (Cartesian) and (Absolute) are selected on the Keyboard Input bar.

2. On the View toolbar, click .

This option updates the display of the model to fit inside the View window.

3. On the Draw toolbar, click .

4. In the Keyboard Input bar, enter the following coordinates to draw the plate.

Start coordinates	0, 35	Press ENTER
End coordinates	0, 55	Press ENTER
End coordinates	30, 55	Press ENTER
End coordinates	30, 0	Press ENTER
End coordinates	0, 0	Press ENTER
End coordinates	0, 5	Press ENTER
End coordinates	20, 5	Press ENTER
End coordinates	20, 35	Press ENTER
End coordinates	0, 35	Press ENTER

5. Press ESC.

4.2 Make the plate component

A new material will be added to the material library: aluminum with a conductivity of 0.3278e8.

1. On the Selection toolbar, click  (Select Construction Slice Surfaces tool).

2. Click the mouse pointer inside the surface of the plate.

3. On the Model toolbar, click  (Make Component in a Line tool).

4. In the **Name** box, enter **Plate**.

5. Click New Material.

For this problem, you will have to create a new material in your material database.

6. On the General page, enter the following data:

- Name: **Aluminum 0.3278e8**

- Display color: *Click Display Color and select an appropriate color*

- Transparency: Optional

- Description: *Optional*

- Categories: *Optional*

7. Click *Next*.

8. On the Options page, select the following:

- Magnetic **Permeability**

- Electric **Conductivity**

- Electric **Permittivity**

9. Using the *Next* button to advance to the appropriate pages, enter the following values:

- Temperature Celsius = **20**

- Relative Permeability = **1**

- Coercivity Amps/m = **0**

- Conductivity Siemens/m = **0.3278e8**

- Relative Permittivity = **1**

10. Once you have entered all the values, click *Finish* in the *Confirmation page* to create the new material.

11. From the *Material* drop down list, select **Aluminum 0.3278e8**.

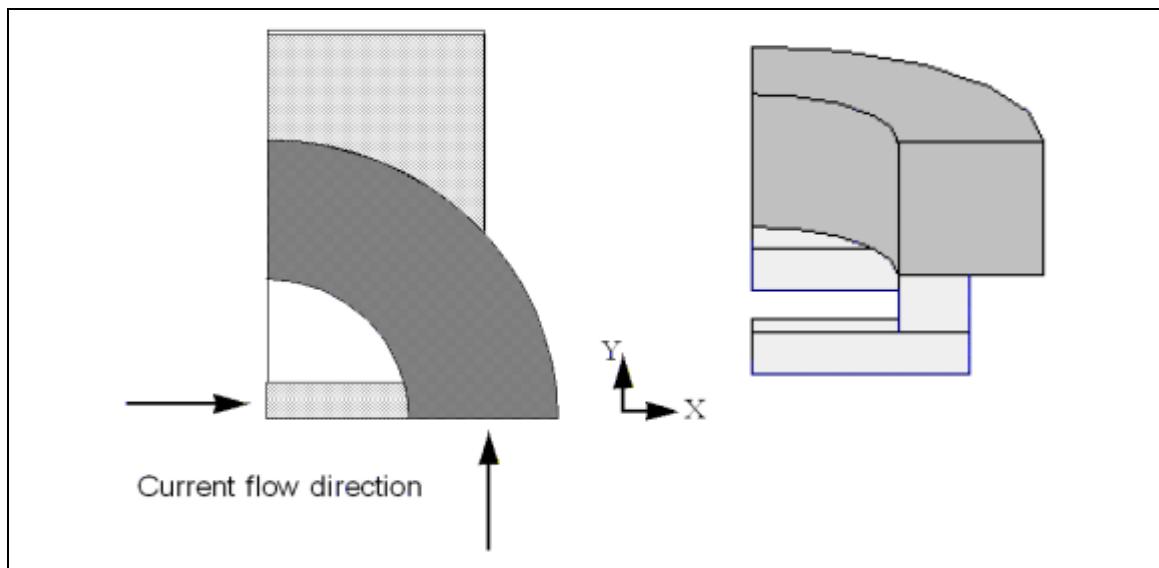
12. In the *Distance* box, enter **6.35**.

13. Click *Ok* to accept the settings.

14. On the File menu, click *Save*.

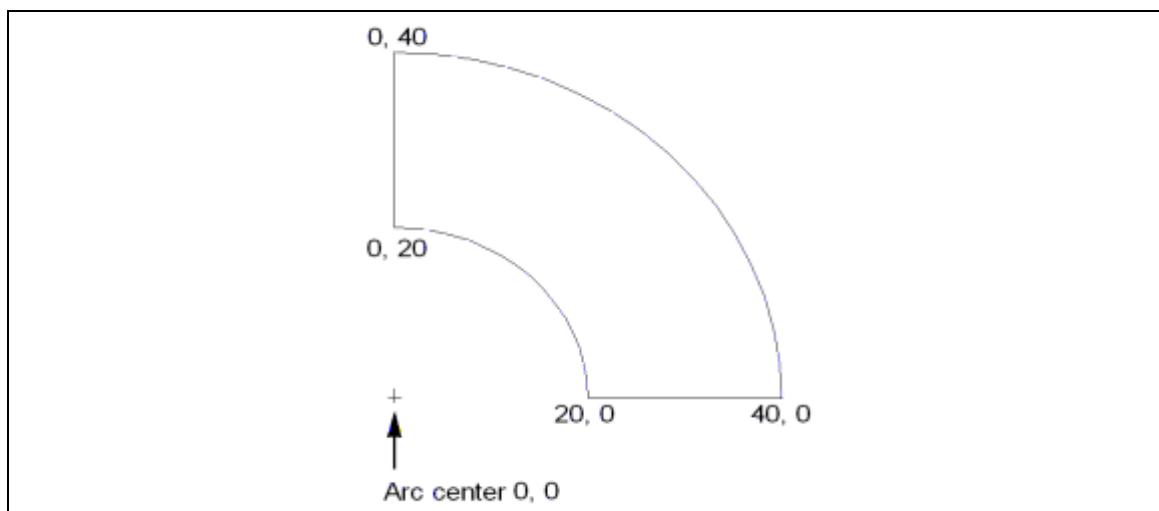
5 Create the excitation

In this model, there is one current-driven coil. The coil is a quarter-circle 15 millimeters above the plate.



6 Create the coil component

The geometry of the current-driven coil is shown below.



6.1 Move the construction slice

The current-driven coil begins 15 millimeters above the end surface of the plate.

1. On the Preset View toolbar, click  [Show XY (+Z)].
2. On the Object page, select *Plate, Face#2 (End Face)*.
3. On the Draw toolbar, click  (Move Construction Slice tool).
4. Make sure the option *To the Currently Selected Surface* is selected.
5. Click *Apply*.
6. In the Move Construction Slice dialog, click *Along A Line*.
7. In the *Distance* box, enter **15**.
8. Click *OK*.

6.2 Delete Construction Slice edges

Since the edges on the construction slice remain even after being swept into a component, we must remove them before proceeding to the next step of the tutorial.

1. On the Edit menu, click *Select Construction Slice Edges*.
2. On the Edit menu, click *Select All*.
3. On the *Standard Tools* toolbar, click  (Delete).

6.3 Draw the geometry

1. On the Draw toolbar, click  (Center, Start, End).
2. In the Keyboard Input bar, enter the following coordinates for **Arc 1**:

Note Arcs are drawn in a counter-clockwise direction.

Center coordinates	0, 0	Press ENTER
Start coordinates	20, 0	Press ENTER
End coordinates	0, 20	Press ENTER

3. In the Keyboard Input bar, enter the following coordinates for **Arc 2**.

Center coordinates	0, 0	Press ENTER
Start coordinates	40, 0	Press ENTER
End coordinates	0, 40	Press ENTER

4. On the Draw toolbar, click the Line drawing tool .
5. In the Keyboard Input bar, enter the following coordinates for **Line 1**.

Start coordinates	20, 0	Press ENTER
End coordinates	40, 0	Press ENTER
		Press ESC.

6. In the Keyboard Input bar, enter the following coordinates for **Line 2**.

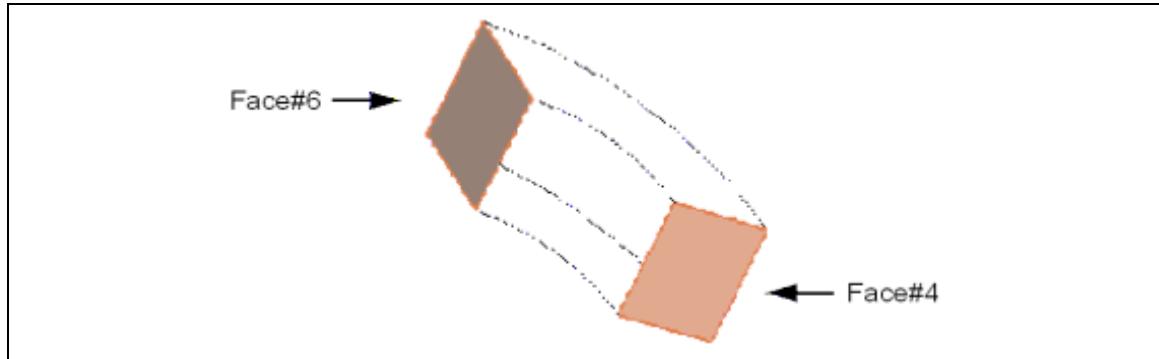
Start coordinates	0, 20	Press ENTER
End coordinates	0, 40	Press ENTER
		Press ESC.

6.4 Make the component

1. On the Edit menu, click Select Construction Slice Surfaces.
2. Click the mouse pointer inside the construction slice surface.
3. On the Model toolbar, click  (Make Component in a Line tool).
4. In the *Name* box, enter **Current component**.
5. In the *Material* drop down list, select **Copper 5.77e7 Siemens/meter**.
6. In the *Distance* box, enter **20**.
7. Click OK to accept the settings.
8. Click Save.

6.5 Make the coil

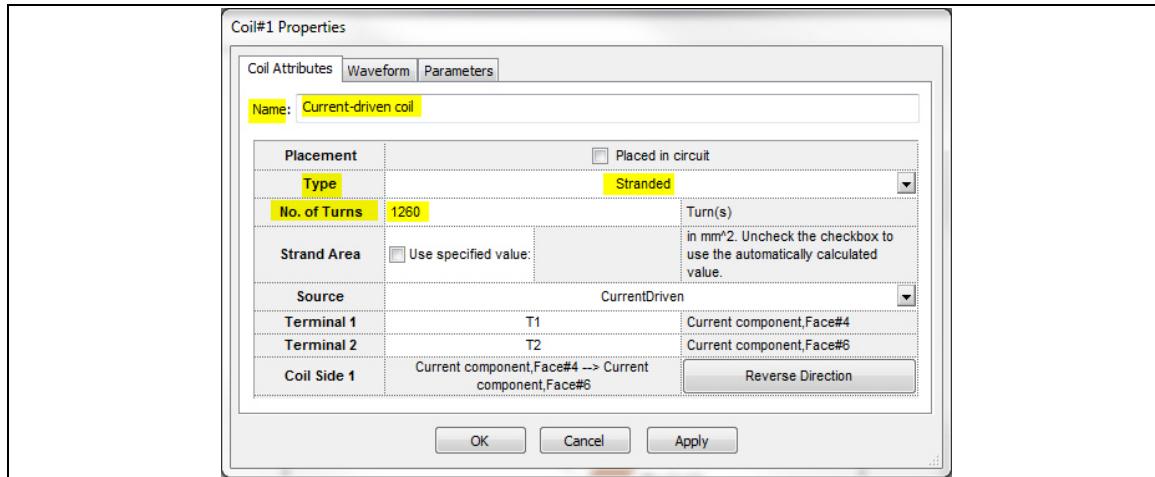
1. On the View toolbar, click  (Automatic View All).
2. On the View toolbar, click  (Examine Model).
3. Holding down the left mouse button, drag the mouse pointer in one or more of the following directions:
 - Drag down to rotate the display downward.
 - Drag up to rotate the display upward.
 - Drag left to rotate the display toward the left.
 - Drag right to rotate the display toward the right.
4. Release the mouse button once the display is rotated about the center of the model.



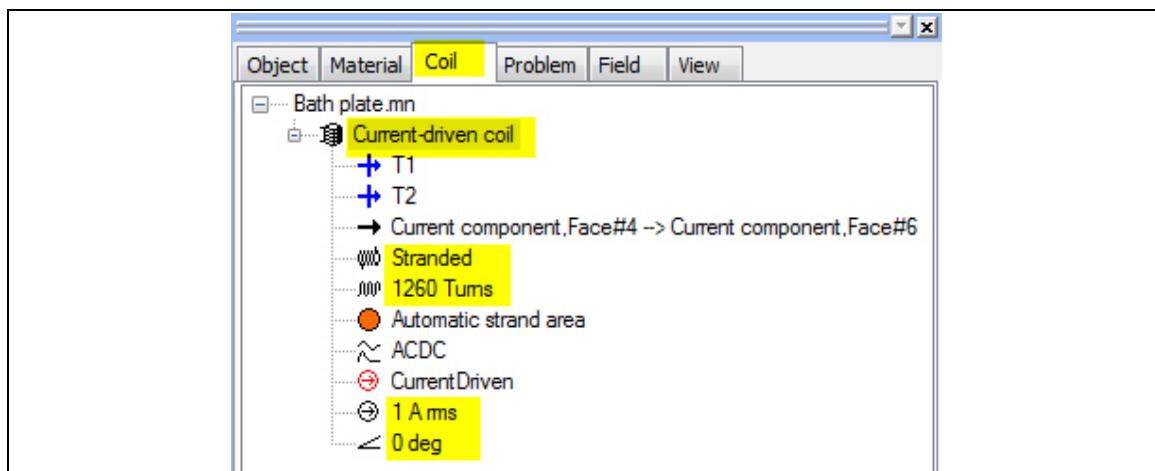
5. On the Object page, keeping the CTRL key pressed, select *Face#4 and Face#6* of the *Current component*.
6. On the Model menu, click *Make Multi-Terminal Coil*.
The coil is listed in the Object page as Coil#1.

6.6 Edit the properties of the coil

1. On the Coil page, select *Coil#1*, right-click and then select *Properties*.
The Coil Properties' Coil Attributes page appears.



2. Do the following:
 - In the *Name* box, enter **Current-driven coil**.
 - In the *Type* dropdown list, select **Stranded**.
 - In the *No. of Turns* text box, enter **1260**.
3. Select the Waveform tab and enter the following values for DC:
Amplitude: **1**
The current is 1 ampere per turn. The phase is 0 degrees.
4. Click **OK**.
The properties of the coil are updated on the Coil page.



5. Click *Save*.

7 Modify the mesh

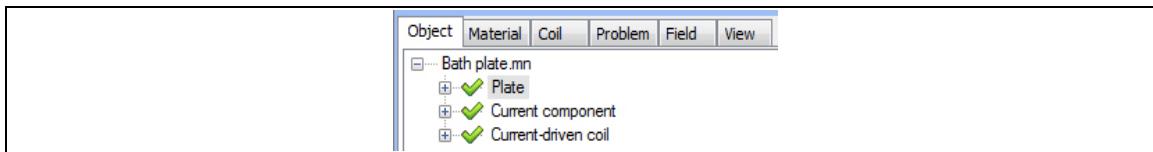
In the 3D finite element method of analysis, the model is divided into a mesh of tetrahedral-shaped elements. Each element is defined by four vertices (nodes). The vector field inside each element is represented by a polynomial with unknown coefficients. The finite element analysis is the solution of the set of equations for the unknown coefficients.

The accuracy of the solution depends upon the nature of the field and the size of the mesh elements. In regions where the direction or magnitude of the field is changing rapidly, high accuracy requires small elements. One method of increasing mesh density is to set the *maximum element size* for a component volume or specific faces of a component. The following procedure will demonstrate this method.

A *maximum element size* modification of 5.5 and 8.5 millimeters will be applied to the *Plate* and *Current component*, respectively.

7.1 Set the maximum element size for each component

1. In the Object page of the Project bar, begin by selecting the *Plate* component.

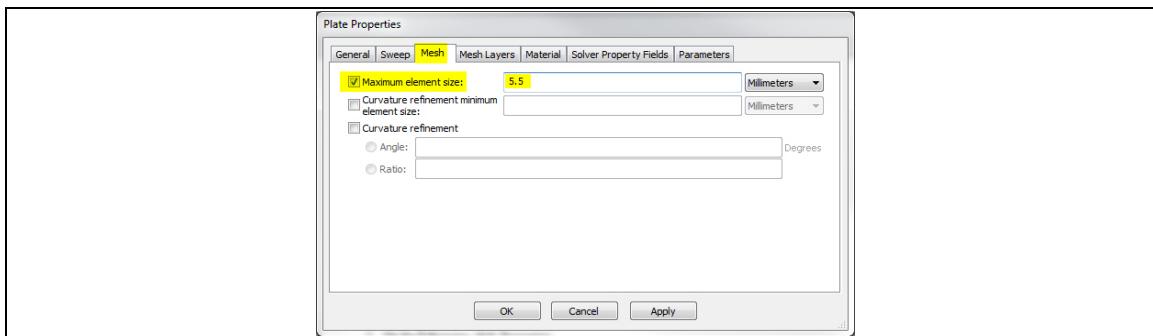


2. On the Edit menu, click *Properties*.

The Properties dialog appears.

3. Select the *Mesh* tab.

4. Click inside the *Maximum element size* checkbox, and then type **5.5** in the text box.



5. Click *Apply*.

Tip Clicking *Apply*, instead of *OK*, keeps the dialog open and allows us to proceed to the next component without having to repeat steps 2 and 3.

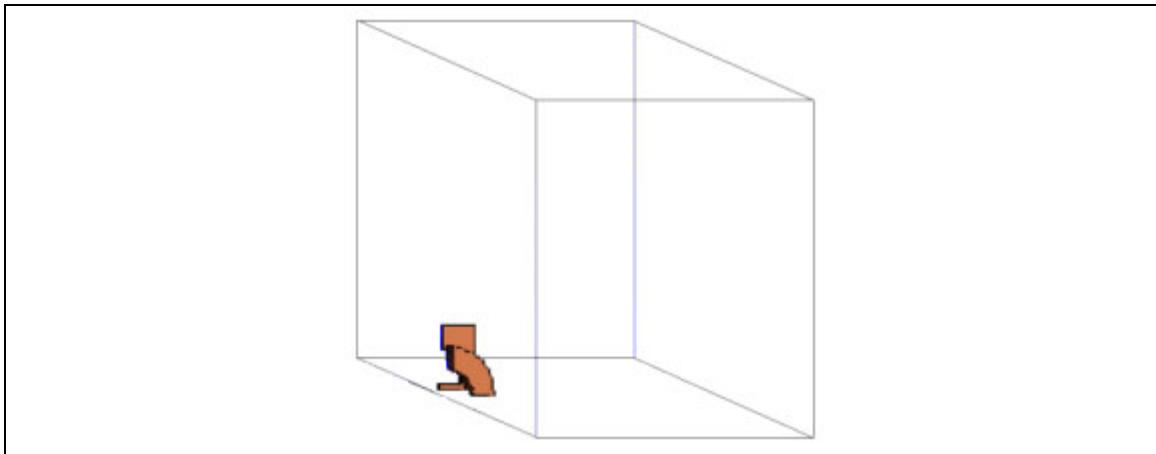
6. In the Object page, select *Current component*.

Notice that the text in the Properties dialog Title Bar has now changed to read *Current component Properties*.

7. Click inside the *Maximum element size* checkbox, and then type **8.5** in the text box.
8. Click *OK*.

8 Add a boundary condition

In this model, an air box is added around the Bath plate. The default boundary condition, tangential flux, is automatically applied to the outermost surfaces of the air box.



9 Create the air box

The starting surface of the air box begins 179.5 millimeters below the starting surface of the plate.

9.1 Move the construction slice

1. On the Preset View toolbar, click [Show XY (+Z)].
2. On the Object page, select *Plate, Face#1 (Start Face)*.
3. On the Draw toolbar, click (Move Construction Slice tool).
4. Make sure that To The Currently Selected Surface is selected.
5. Click Apply.

Tip Clicking Apply, and not OK, keeps the dialog open for the next step.

6. Select Along A Line.
7. In the *Distance* box, enter **-179.5**.
8. Click OK.

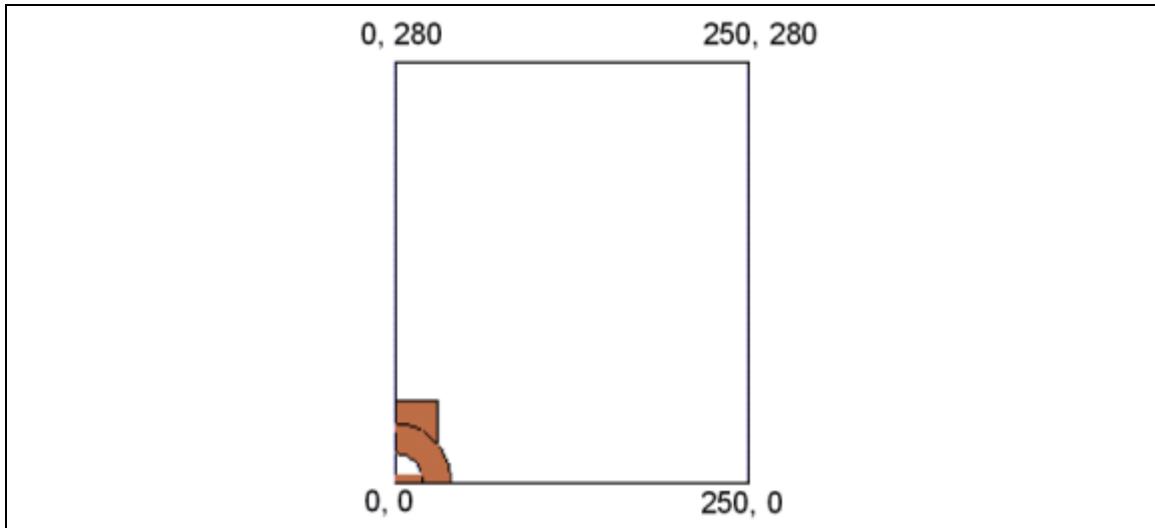
9.2 Delete Construction Slice edges

Since the edges on the construction slice remain even after being swept into a component, we must remove them before proceeding to the next step of the tutorial.

1. On the Edit menu, click Select Construction Slice Edges.
2. On the Edit menu, click *Select All*.
3. On the *Standard Tools* toolbar, click  (Delete).

9.3 Draw the geometry

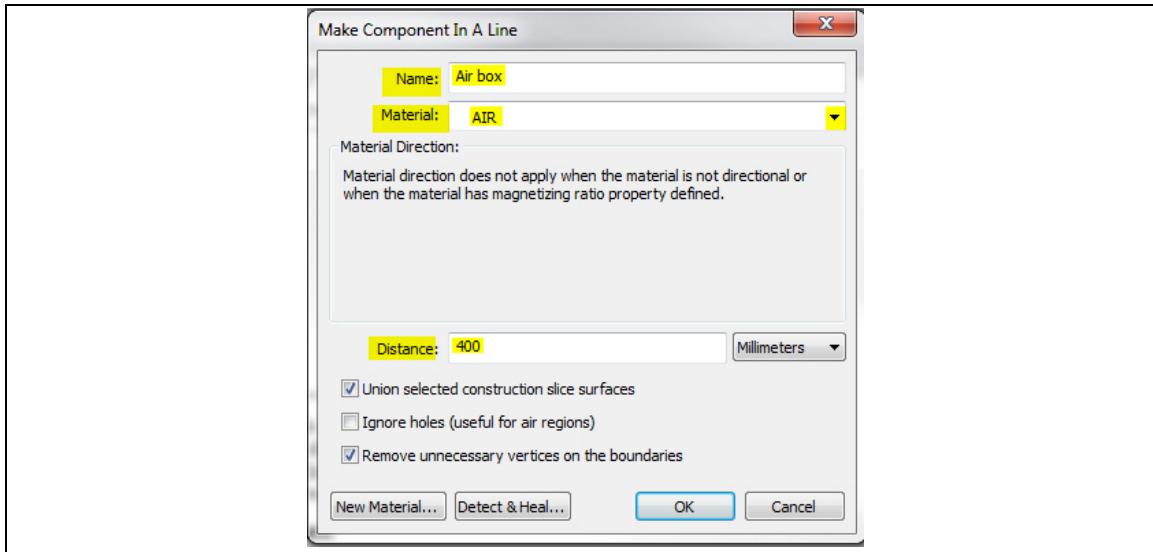
The geometry of the air box is shown below.



1. On the Draw toolbar, click  (Line drawing tool).
2. On the View menu, click *Update Automatically*.
3. From the *Keyboard Input Bar*, draw the air box using the coordinates shown above, pressing ESC after the last coordinates have been entered.

9.4 Make the component

The air box extends 400 millimeters.



1. Make a component in a line from the *surface of the Air box*.
2. Sweep the component **400 millimeters**.
3. Apply the material **AIR**.
4. In the *Name* box, type **Air box**.
5. Click Save.

9.5 Rotate the model

Verify the position of the Bath plate in the air box by rotating the model.

1. On the View toolbar, click (Automatic View All).
2. On the View toolbar, click (Examine Model).
3. Holding down the left mouse button, drag the mouse pointer in one or more of the following directions:
 - Drag down to rotate the display downward.
 - Drag up to rotate the display upward.
 - Drag left to rotate the display toward the left.
 - Drag right to rotate the display toward the right.

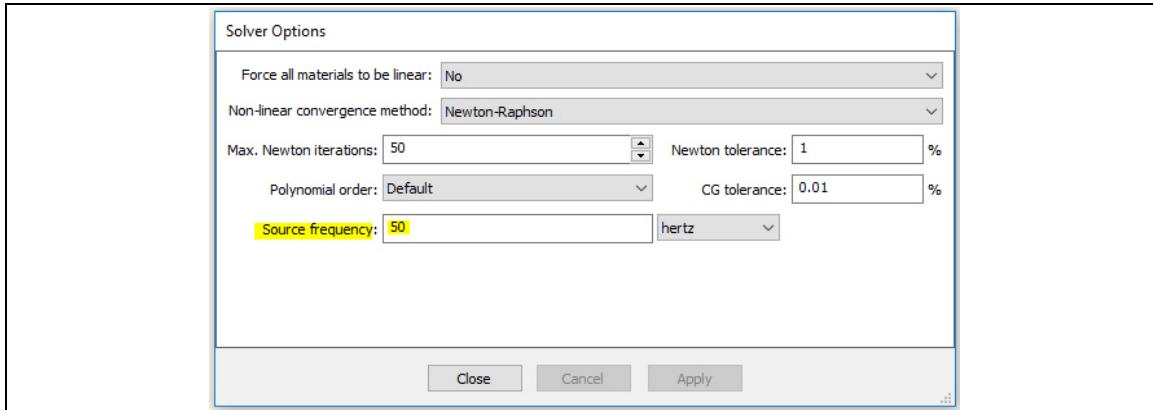
The display is rotated about the center of the model.

Release the mouse button.

10 Set the source frequency

The Bath plate is solved at 50 Hertz.

1. On the Solve menu, click *Set Solver Options*.
The *Solver Options* dialog appears.
2. In the *Source Frequency* text box, enter **50**. Hertz is the default unit.

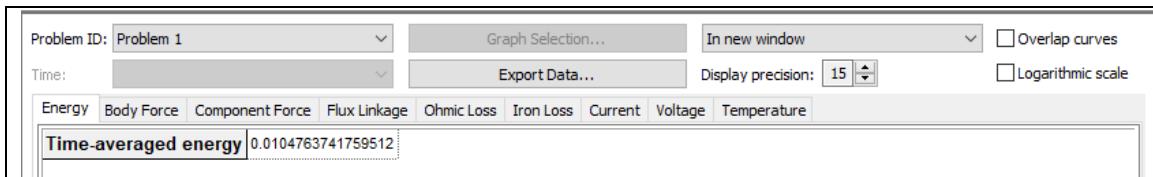


3. Click OK.
4. Click Save.

11 Solve

- On the Solve menu, click *Time-harmonic 3D*.

The *Time-harmonic 3D Solver Progress* dialog appears briefly and the Results window opens.



12 View the solution results

In this section, the following results are viewed:

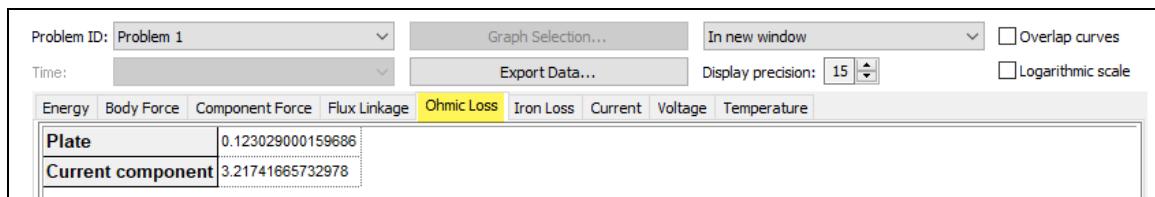
- Time-averaged ohmic loss dissipated in each conductor.
- Plotting Bz at 0', extracted along a contour and graphed.

12.1 View the time-averaged Ohmic loss

The Results window is automatically displayed when the solution is complete.

- Select the *Ohmic Loss* tab.

The *Ohmic Loss* page displays the time-averaged Ohmic loss in each conducting component (i.e. the Plate and the Current Component) in the model.



12.2 Obtain Bz at 0' and graph the results

To obtain **Bz at 0'** for graphing, the following steps must be completed:

- Move construction slice.
- Update the field view to display B at 0'.
- Plot Bz at 0' on a graph.

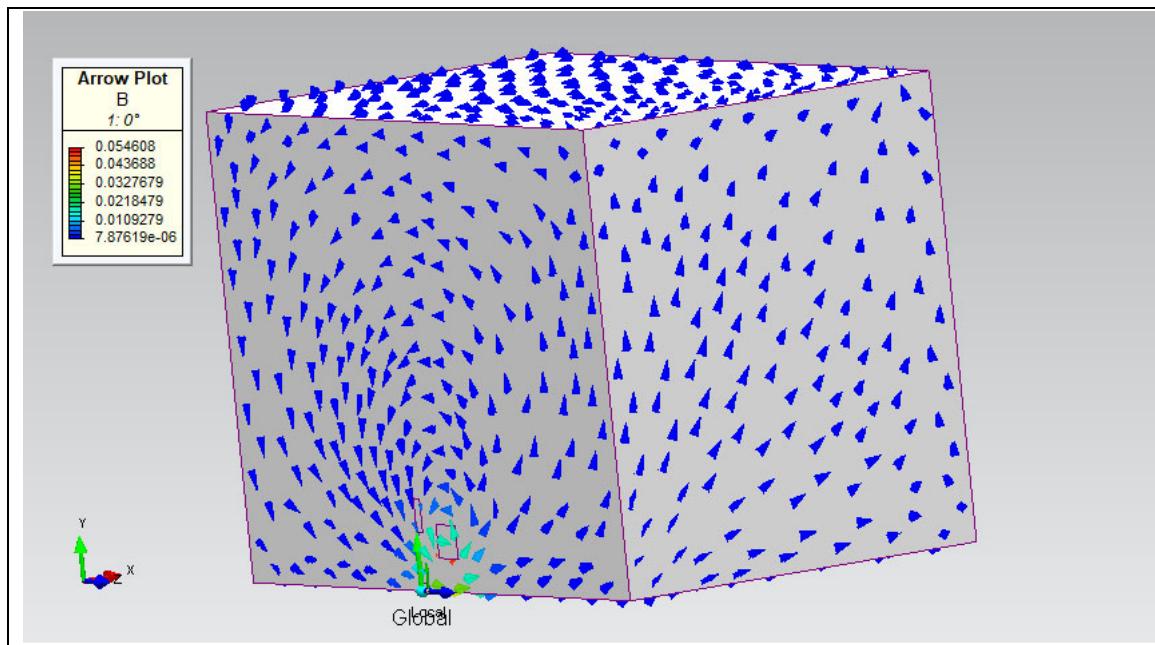
12.3 Move the construction slice

Since this is a 3D problem, we need to move the construction slice to an appropriate location (in this case, 0.5 Millimeters above the plate), from where we can then define the segment to create a graph.

1. Before moving the construction slice, switch back to the View window by clicking the View tab located at the bottom of the window.
2. On the Object page, select *Plate, Face#1 (Start Face)*.
3. On the Draw toolbar, click (Move Construction Slice tool).
4. Make sure that To the Currently Selected Surface is selected.
5. Click Apply.
Tip Clicking Apply, and not OK, keeps the dialog open for the next step.
6. Select Along A Line.
7. In the *Distance* box, enter **6.85** Millimeters.
8. Click OK.

12.4 Update field view to display B at 0°

1. On the Project bar, select the *Field* tab.
2. For both the *Contour* and *Shaded* plots (click the tabs at the bottom of the Field page), in the *Fields to Display* list, select **None**.
3. Select the *Arrow* tab, and then in the *Fields to Display* list, select **B at 0°**.
4. At the bottom of the Field page, press *Update View*.



12.5 Plotting Bz at 0° on a graph

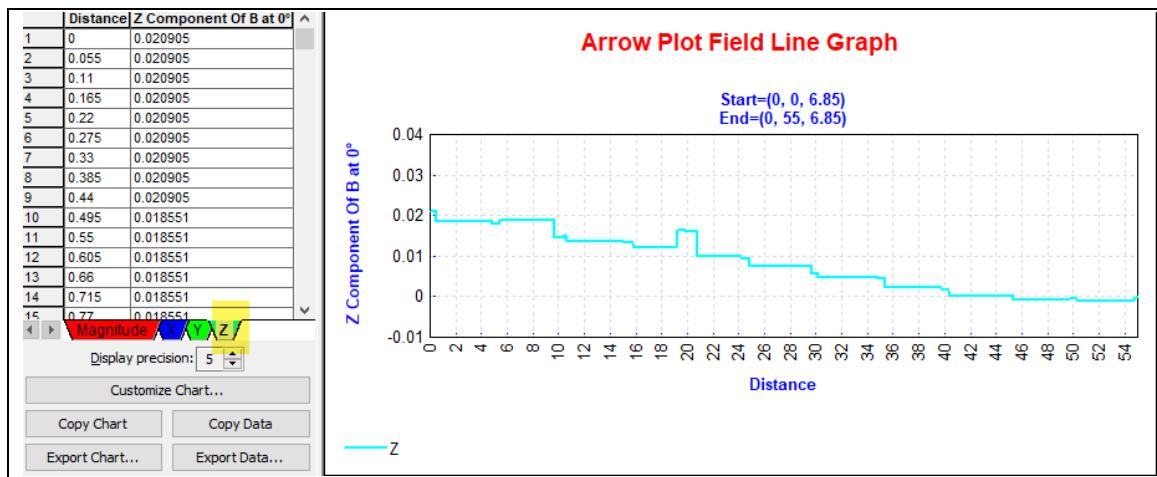
In this procedure we are going to use the Simcenter MAGNET post-processing feature (Field Line Graph) that allows us to define a line segment, extract the 1000 field values along that segment, and then plot the field quantities on a graph.

Note Although not shown in this procedure, *Field Arc Graph* and *Field Circle Graph* are also available for post-processing. Please refer to the Help for more information.

1. On the Tools menu, click *Field Line Graph*.
2. In the Keyboard Input bar, enter the coordinates **(0, 0)** and **(0, 55)**, pressing Enter after each set of numbers.

Note If you prefer, you can also use the mouse to draw the line graphically.

3. The graph is automatically displayed in a new View window. Select the **Z tab** to view the *Z Component of B at 0'* graph.



12.6 Save the model

You have now completed the Bath plate tutorial.

1. On the File menu, click *Save*.
2. On the File menu, click *Close*.

13 Summary

In this tutorial, you built and solved the Bath plate. The skills you learned include:

- Creating a current-driven coil
- Setting the source frequency
- Viewing the time-averaged Ohmic loss.
- Obtaining and creating a graph of Bz at 0' along the contour

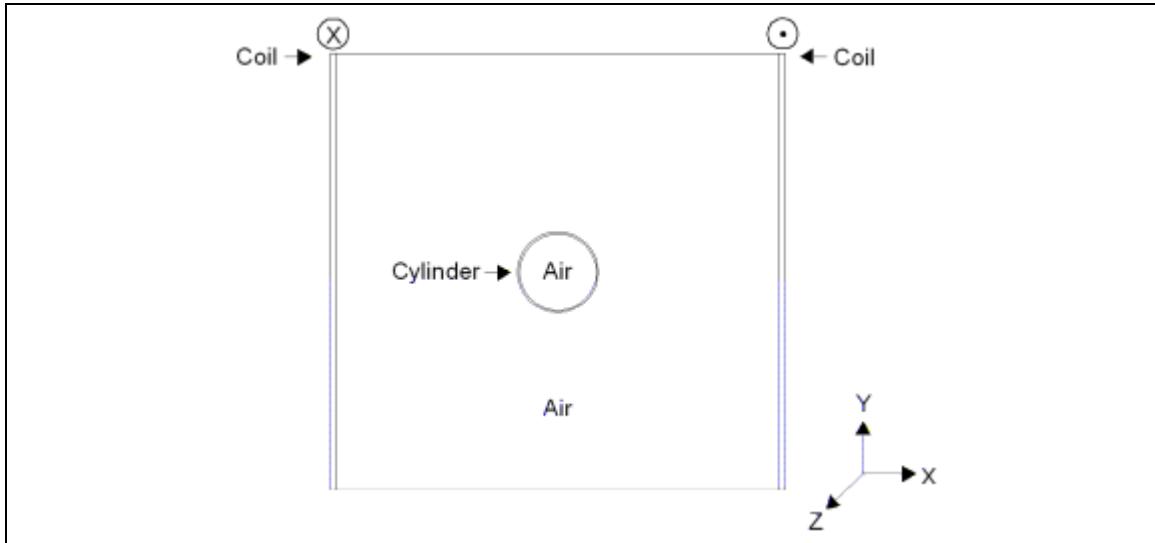
Tutorial #9

3D Transient

Felix short cylinder

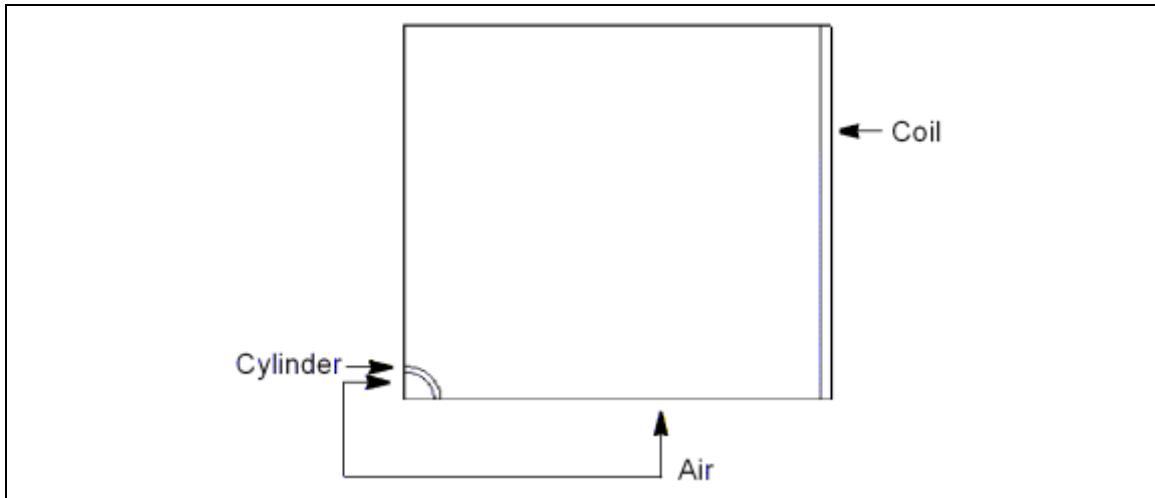
1 Modeling plan

The problem is comprised of a hollow conducting cylinder lying in a uniform field. An infinite length stranded coil lying on either side of the cylinder provides the uniform field. The magnetic field of the problem is perpendicular to the cylinder's axis. The magnetic field decays exponentially in time.

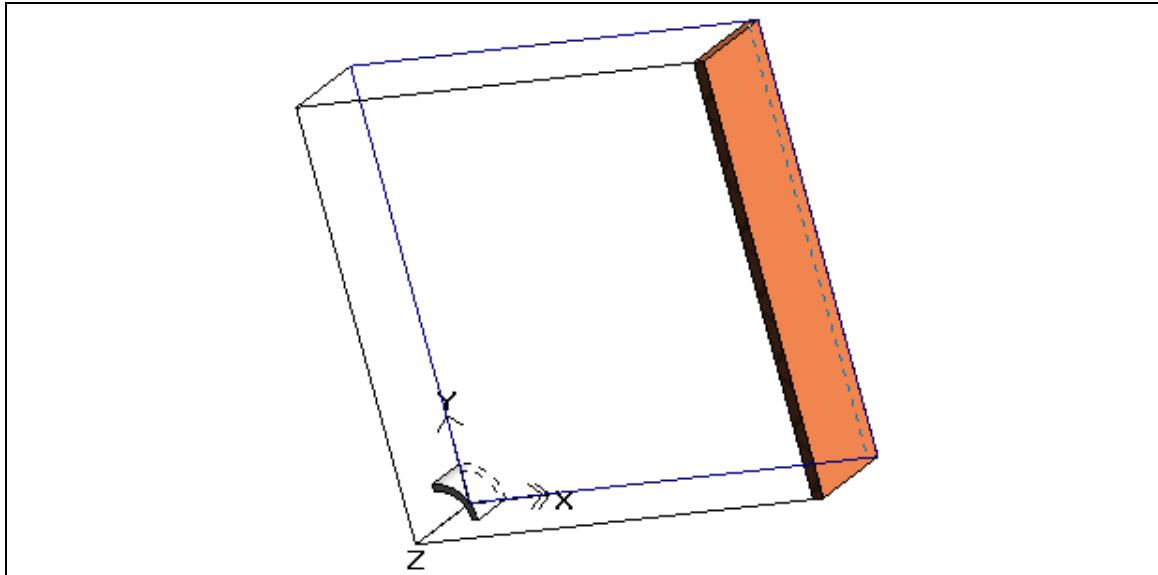


After solving, the shaded plot is viewed and animated. Once the animation is complete, the stored energy of the system is graphed. Both the animation and the graph are viewed through all the time steps.

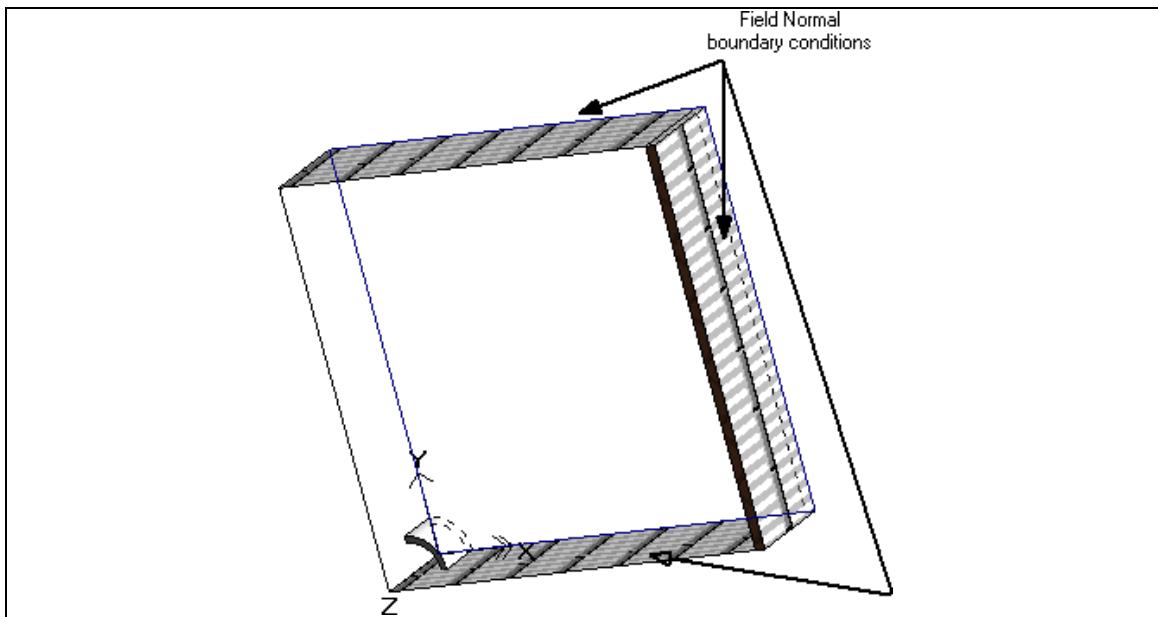
Symmetry conditions allow for only one-eighth of the problem to be modeled. The model is built from three components: a quarter-cylinder, a coil, and an air space that surrounds the two other components. The geometry of the model is shown below.



After drawing, the geometry is swept into components and a coil is created from one of the components.



Boundary conditions are applied to indicate symmetry. The Field Normal boundary condition is applied to the top, bottom, and right surfaces of the air space. The default boundary condition, Flux Tangential, is automatically applied to the remaining surfaces.



Tip The Field Normal boundary condition on the right surface of the air box (representing the outside of the coil) forces the coil flux outside to infinity. If the surface had the Flux Tangential boundary condition, the coil flux would be forced to return inside the coil (which would give incorrect field values inside the coil).

The density of the mesh will be increased in the area of the cylinder to improve solution accuracy.

2 Open a new model

1. Start Simcenter MAGNET.
The Main window appears.
2. If Simcenter MAGNET is already running, select *New* from the File menu to open a new model.
If you have already used Simcenter MAGNET, the window displays the settings that were last active.

To maximize the window, click  on the top right corner of the Simcenter MAGNET Main window.

2.1 Name the model

1. On the File menu, click *Save As*.
2. In the *Save As* dialog box, enter **Felix short cylinder** as the name of the model.
3. Choose the drive and directory where you want to place the model.
4. Click *Save As*.

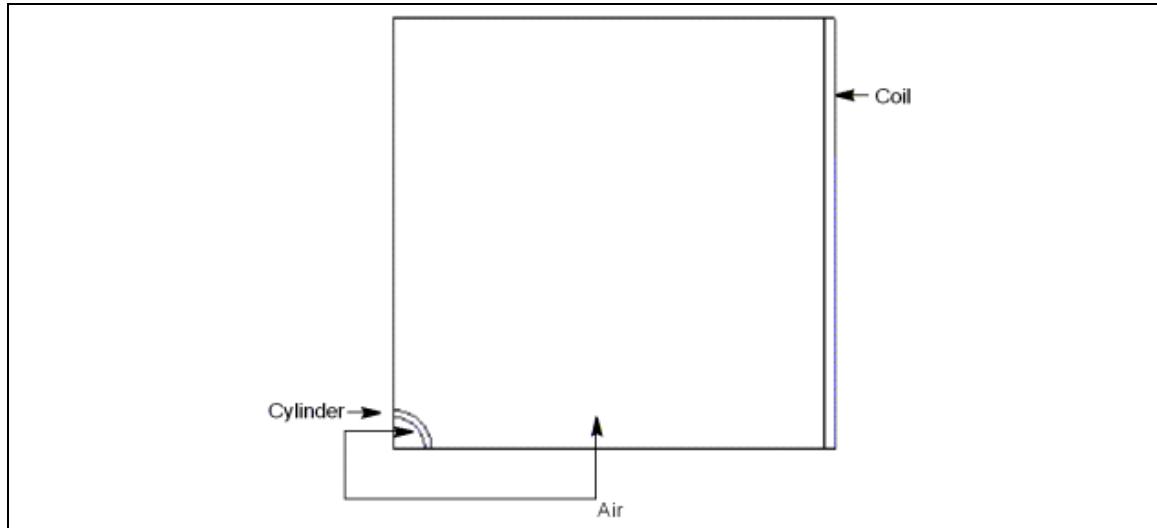
2.2 Set the preferred unit for time

The Simcenter MAGNET default unit for time is milliseconds. The Felix short cylinder time steps will be measured in seconds. You can set seconds to be preferred unit for time in all the Simcenter MAGNET dialogs. This option is set in the Model property page.

1. In the Object page of the Project bar, select the model (i.e. *Felix short cylinder.mn*).
2. On the Edit menu, click *Properties*.
The Properties dialog appears.
3. On the Units property page, in the *Time* drop down list, select **Seconds**.
4. Click OK.

3 Build the geometric model

The geometry of the model is shown below.



3.1 Set the drawing area

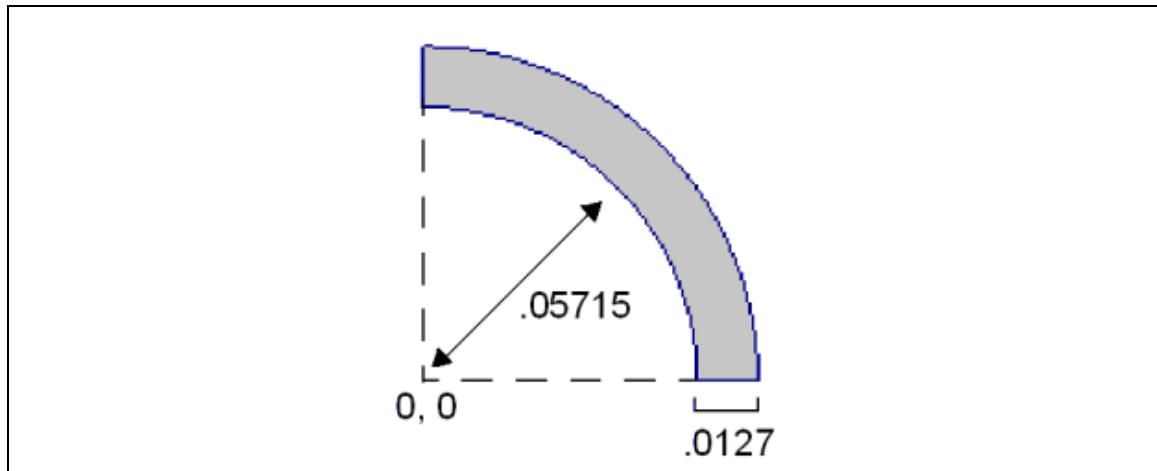
The model will be drawn using the Keyboard Input bar, so the construction grid is not needed and can be removed from the display. The drawing unit is meters (the Simcenter MAGNET default).

- Turn off the display of the Construction Grid (if it is displayed).

If the grid is visible, toggle *Construction Grid* on the View menu.

3.2 Create the cylinder component

The quarter-cylinder has a wall thickness of .0127 meters and an inner radius of 0.05715 meters.



3.3 Draw the cylinder

1. If the Keyboard Input bar is not already displayed at the bottom of the Main window, select *Keyboard Input Bar* on the Tools menu.



2. Make sure that (Cartesian) and (Absolute) are selected on the Keyboard Input bar.
3. On the Draw toolbar, click .
4. In the Keyboard Input bar, enter the following coordinates for the inner arc of the cylinder.
Note Arcs are drawn in a counter-clockwise direction.

Center coordinates 0, 0 Press ENTER

Start coordinates 0.05715, 0 Press ENTER

End coordinates 0, 0.05715 Press ENTER

5. In the Keyboard Input bar, enter the following coordinates for the outer arc of the cylinder.

Center coordinates 0, 0 Press ENTER

Start coordinates 0.06985, 0 Press ENTER

End coordinates 0, 0.06985 Press ENTER

6. On the Draw toolbar, click .

7. In the Keyboard Input bar, enter the following coordinates to draw the lines of the cylinder.

Start coordinates 0, 0.05715 Press ENTER

End coordinates 0, 0.06985 Press ENTER

Press ESC

8. In the Keyboard Input bar, enter the following coordinates to draw the lines of the cylinder.

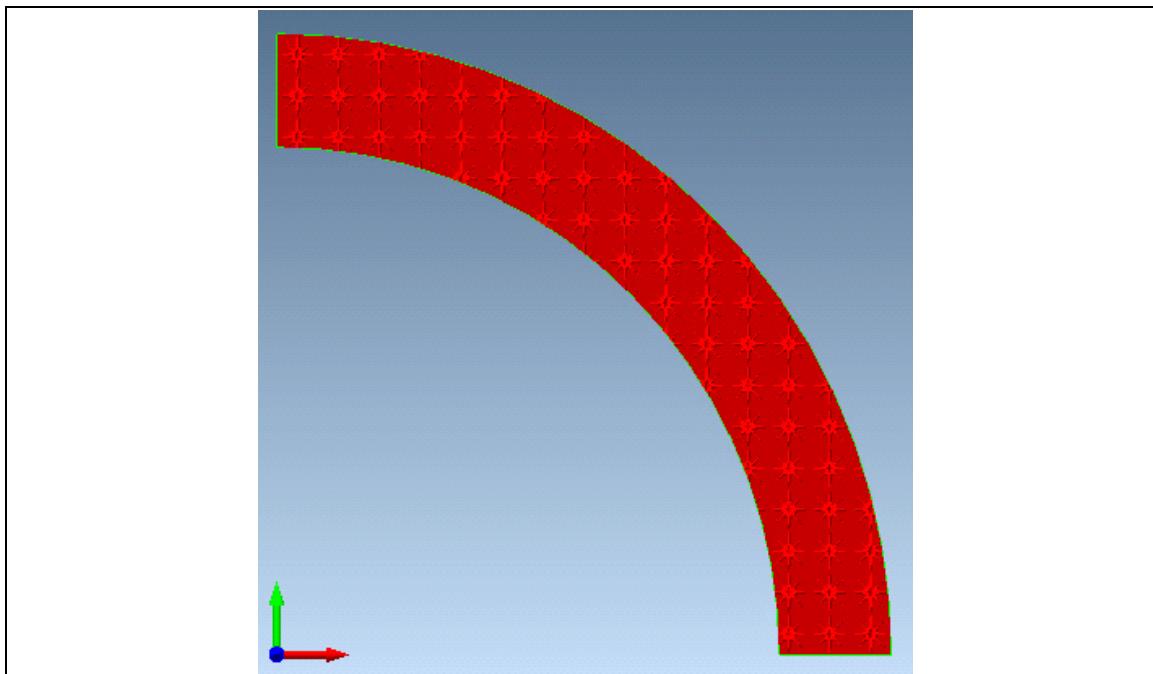
Start coordinates 0.05715, 0 Press ENTER

End coordinates 0.06985, 0 Press ENTER

Press ESC

3.4 Make the component of the cylinder

1. On the Edit menu, click Select Construction Slice Surfaces.
2. Click the mouse pointer inside the surface of the cylinder.
The surface is highlighted when selected.



3. On the Model toolbar, click  (Make Component in a Line tool).
4. In the Name box, enter **Cylinder**.

Note If you have already created the material called *Aluminum 6061* in Tutorial #1 (Cylindrical Shield – 2D Time-Harmonic tutorial) or in Tutorial #5 (Spherical Shield – 2D Time-Harmonic tutorial), please proceed to step 12. If not, proceed to step 5.

5. Click New Material.

For this problem, you will have to create a new material in your material database.

Note If you have already created the material called *Aluminum 6061* in Tutorial #1 (Cylindrical Shield – 2D Time-Harmonic tutorial) or in Tutorial #5 (Spherical Shield – 2D Time-Harmonic tutorial), please proceed to step 12.

6. On the General page, enter the following data:

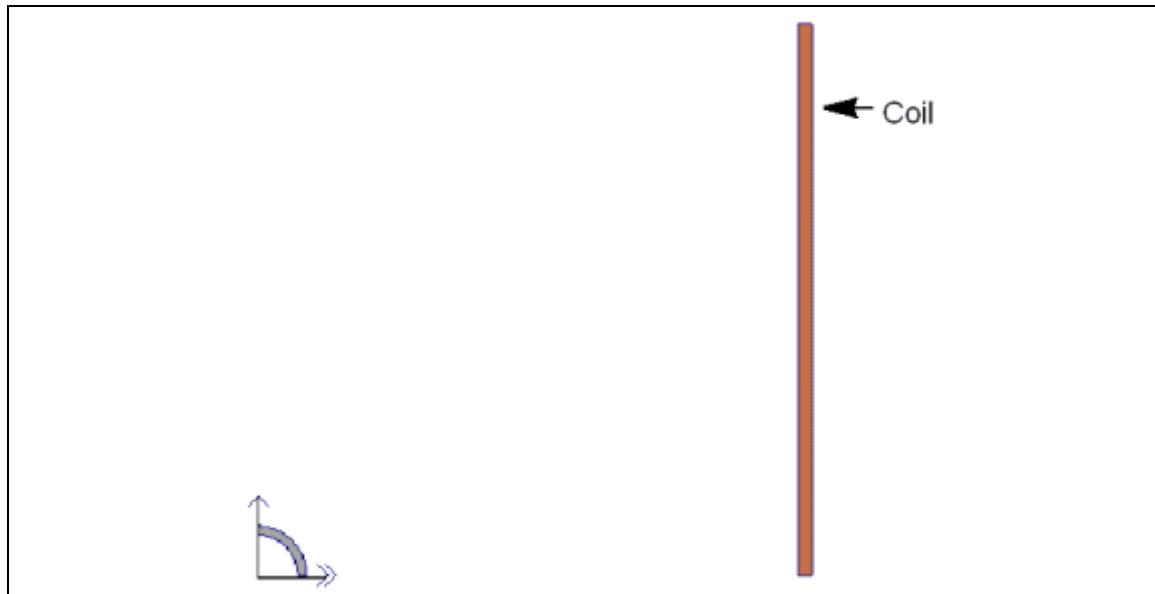
- Name: **Aluminum 6061**
- Display color: *Click Display Color and select an appropriate color*
- Transparency: *Optional*
- Description: *Optional*
- Categories: *Optional*

7. Click *Next*.

8. On the *Options page*, select the following:
 - Magnetic Permeability
 - Electric Conductivity
9. Using the *Next* button to advance to the appropriate pages, enter the following values:
 - Temperature Celsius = **20**
 - Relative Permeability = **1**
 - Coercivity Amps/m = **0**
 - Conductivity Siemens/m = **2.538e7**
10. Once you have entered all the values, click *Finish* in the *Confirmation page* to create the new material.
11. From the *Material* drop down list, select **Aluminum 6061**.
12. In the *Distance* box, enter **0.1**.
13. Click *OK* to accept the settings.
14. On the File menu, click *Save*.

4 Create the coil component

The geometry of the coil component is shown below.



4.1 Draw the coil component

1. On the Draw toolbar, click  (Line drawing tool).
2. On the View toolbar, click  (Update Automatically).
3. In the Keyboard Input bar, enter the following coordinates to draw the coil.

Start coordinates	0.8, 0	Press ENTER
End coordinates	0.8, 0.78	Press ENTER
End coordinates	0.78, 0.78	Press ENTER
End coordinates	0.78, 0	Press ENTER
End coordinates	0.8, 0	Press ENTER
4. Press ESC.

4.2 Make the component of the coil

1. On the Edit menu, click Select Construction Slice Surfaces.
2. Click the mouse pointer inside the surface of the coil.
3. On the Model toolbar, click  (Make Component in a Line tool).
4. In the **Name** box, type **Coil component**.
5. In the *Material* drop down list, select **Copper: 5.77e7 Siemens/meter**.
6. In the *Distance* box, type **0.2**.
7. Click OK to accept the settings.
8. On the File menu, click *Save*.

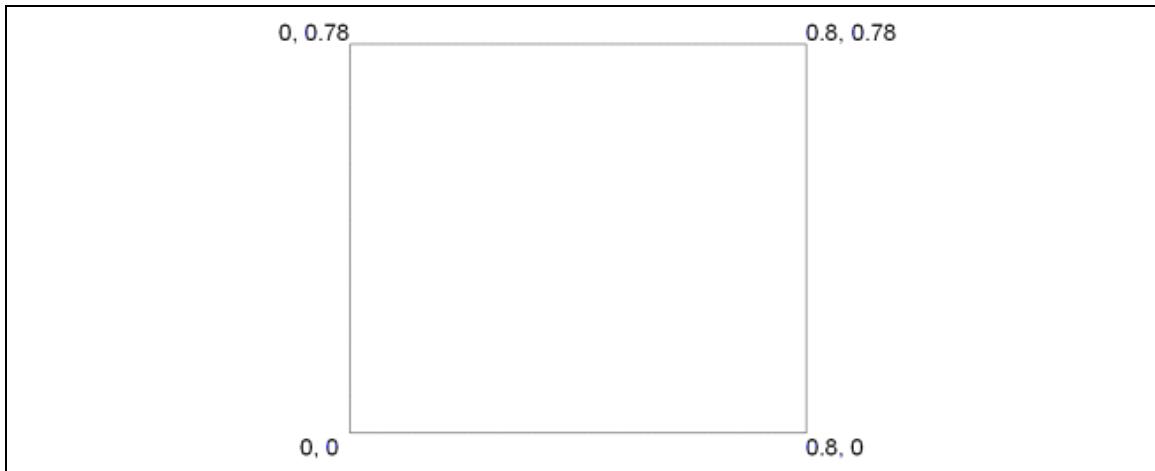
4.3 Delete Construction Slice edges

Since the edges on the construction slice remain even after being swept into a component, we must remove them before proceeding to the next step of the tutorial.

1. On the Edit menu, click Select Construction Slice Edges.
2. On the Edit menu, click *Select All*.
3. On the Standard Tools toolbar, click  (Delete).

5 Create the air box component

The geometry of the air box is shown in the diagram below.



5.1 Draw the geometry of the air box

1. On the Draw toolbar, click (Line Drawing tool).
2. On the View toolbar, click (Update Automatically).
3. In the Keyboard Input bar, enter the following coordinates to draw the air box.

Start coordinates	0, 0	Press ENTER
End coordinates	0.8, 0	Press ENTER
End coordinates	0.8, 0.78	Press ENTER
End coordinates	0, 0.78	Press ENTER
End coordinates	0, 0	Press ENTER
4. Press ESC to stop drawing.

5.2 Make the component of the air box

1. On the Edit menu, click Select Construction Slice Surfaces.
2. Click the mouse pointer inside the surface of the air box.
The surface is highlighted when selected.
3. On the Model toolbar, click (Make Component in a Line tool).
4. In the Name box, enter **Air box**.
5. In the Material drop down list, select **AIR**.
6. In the Distance box, type **0.2**.
7. Click OK to accept the settings.
8. On the File menu, click **Save**.

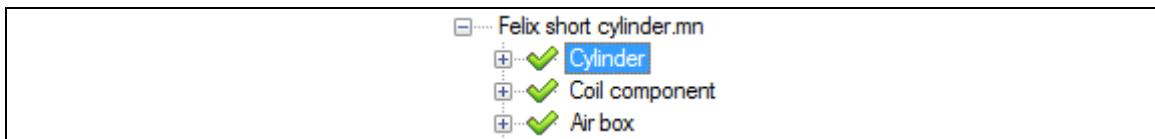
6 Modify the mesh

In the 3D finite element method of analysis, the model is divided into a mesh of tetrahedral-shaped elements. Each element is defined by four vertices (nodes). The vector field inside each element is represented by a polynomial with unknown coefficients. The finite element analysis is the solution of the set of equations for the unknown coefficients.

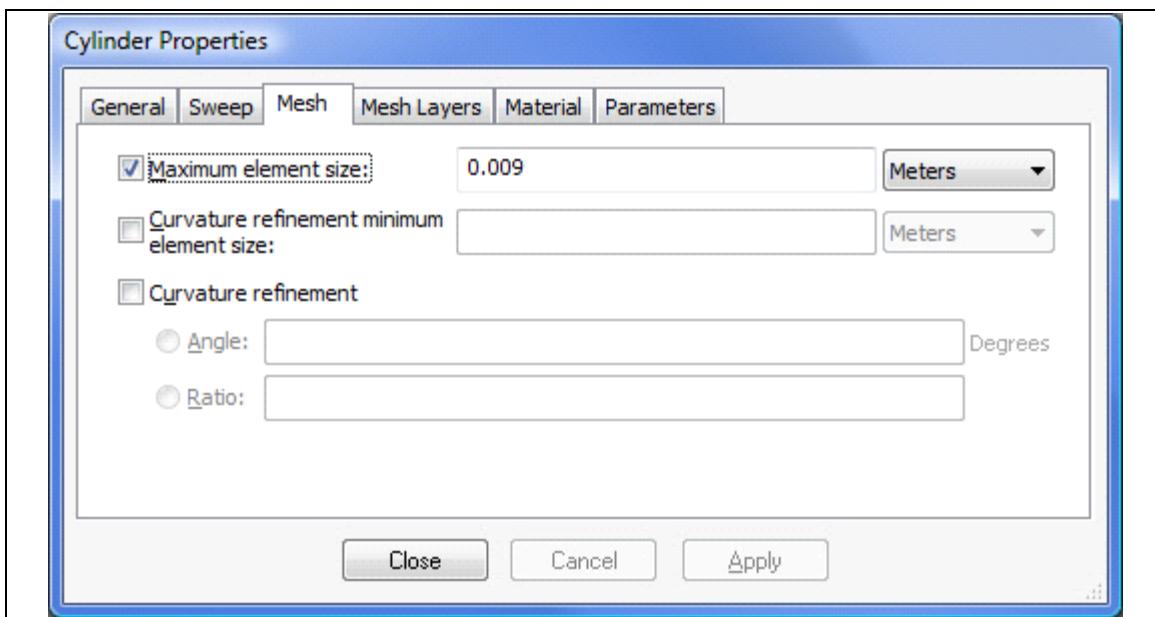
The accuracy of the solution depends upon the nature of the field and the size of the mesh elements. In regions where the direction or magnitude of the field is changing rapidly, high accuracy requires small elements. One method of increasing mesh density is to set the *maximum element size* for a component volume or specific faces of a component. The following procedure will demonstrate this method.

6.1 Set the maximum element size

1. In the Object page of the Project bar, select the Cylinder component.



2. On the Edit menu, click *Properties*.
The Properties dialog appears.
3. Select the *Mesh* tab.
4. Click inside the *Maximum element size* checkbox, and then type **0.009** in the text box.



5. Click OK.

7 Define boundary conditions

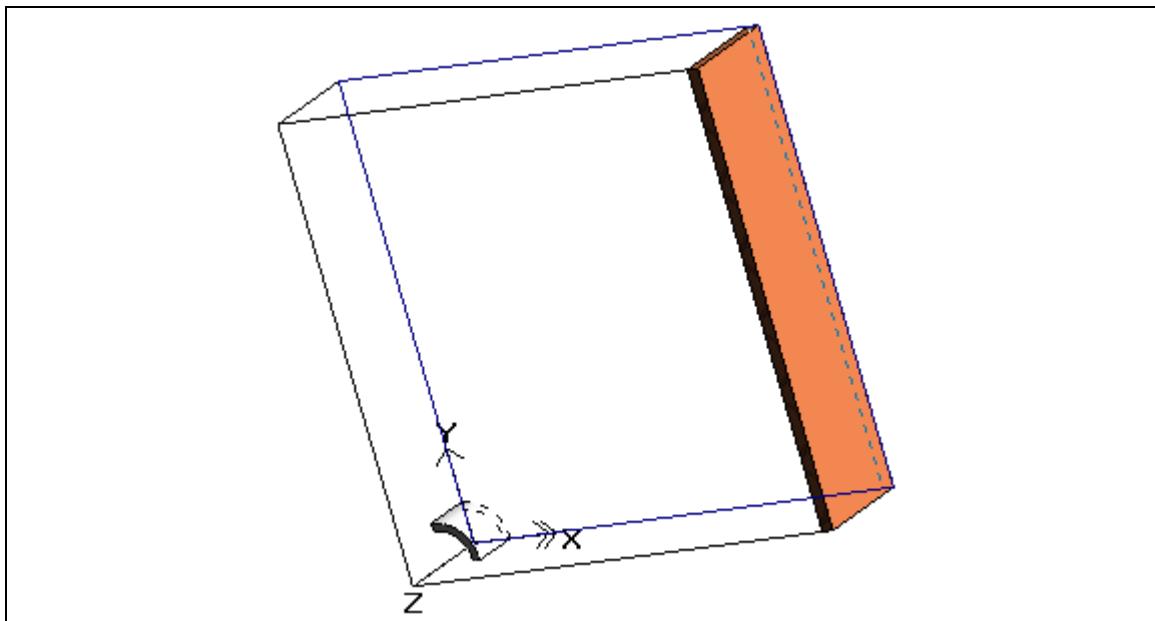
The Field Normal boundary condition is applied to three surfaces of the air box: the top, bottom, and right surfaces. The default boundary condition, Flux Tangential, is automatically applied to the remaining surfaces.

The Field Normal boundary condition constrains to zero the tangential component of the field. The field is made normal (perpendicular) to the boundary. The Flux Tangential boundary condition constrains to zero the normal component of the magnetic flux density. The flux is made to flow tangential to (alongside) the boundary.

7.1 Apply the Field Normal boundary condition

The Object Page of the Project bar lists all of the objects of the model. You can select objects using the Object page.

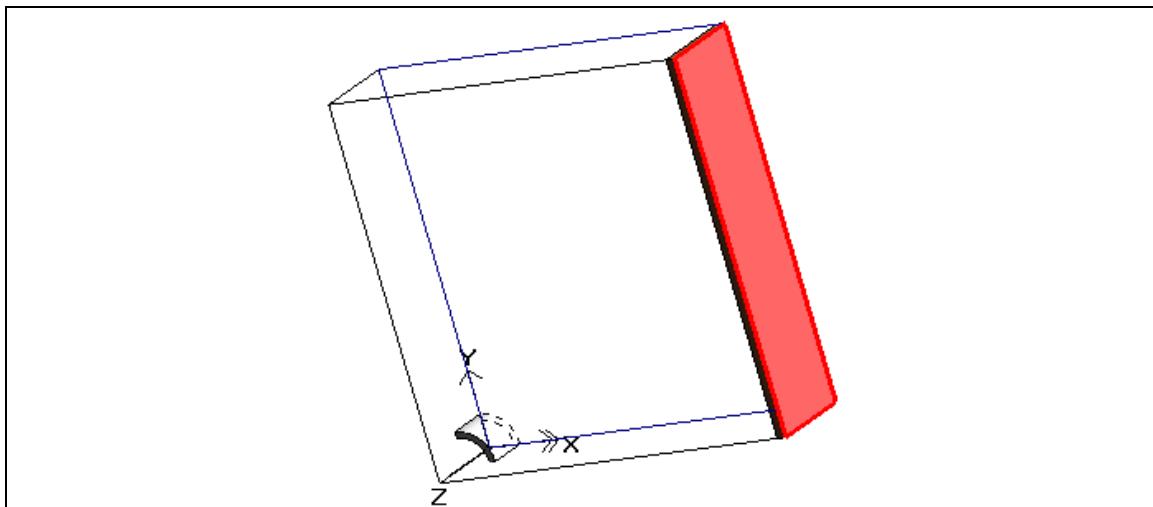
1. On the View toolbar, click  (Examine Model).
2. Holding down the left mouse button, drag the mouse pointer in one or more of the following directions:
 - Drag down to rotate the display downward.
 - Drag up to rotate the display upward.
 - Drag left to rotate the display toward the left.
 - Drag right to rotate the display toward the right.
3. Rotate the model to a 3D view (similar to the diagram below). This rotation displays the surfaces to which the Field Normal boundary condition will be applied.



- In the Object page, click the plus sign (+) beside Air box.
- The surfaces of the component are displayed.
- Click *Face#4*.

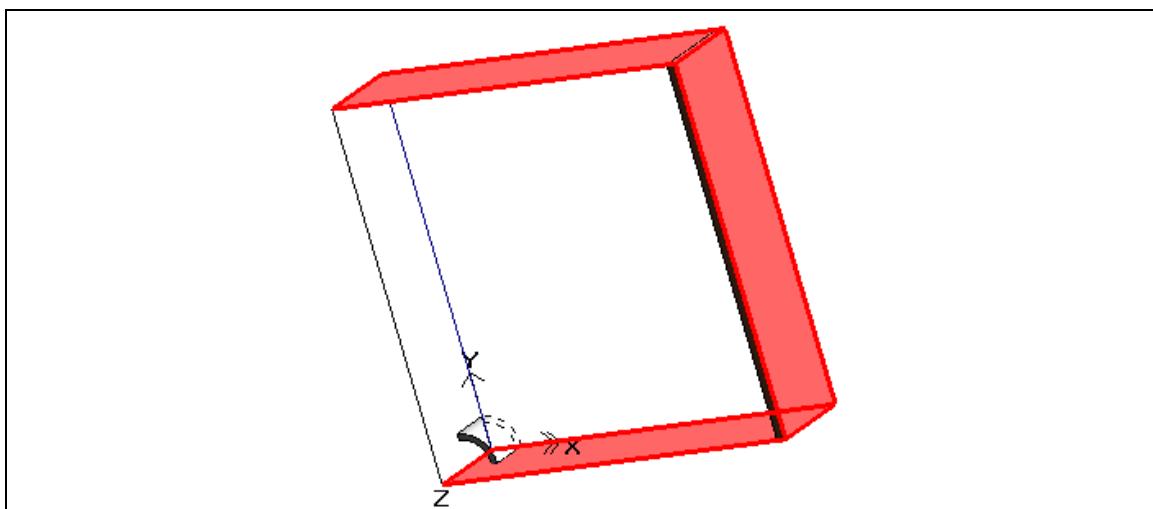


The surface is selected in the View window.



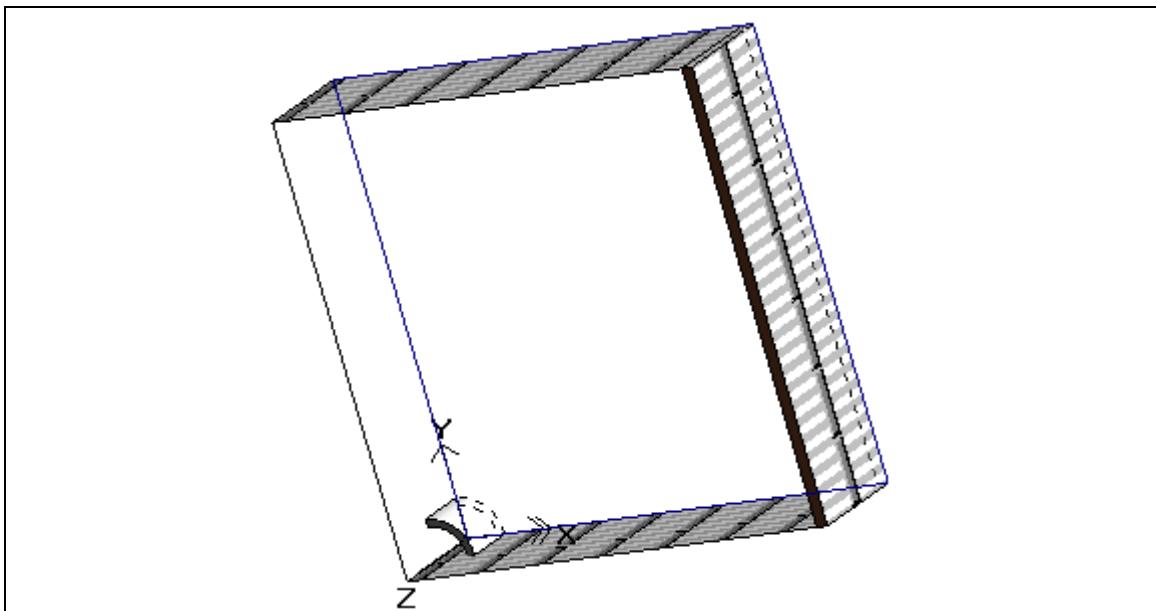
- While holding down the CTRL key on your computer keyboard, also select *Face#3* and *Face#5* of the *Air box* component.

The surfaces are selected in the View window.



Tip You can select multiple objects by holding down the SHIFT or CTRL keys while clicking on additional objects. If you press CTRL while clicking on objects, you can click on an object a second time to de-select it.

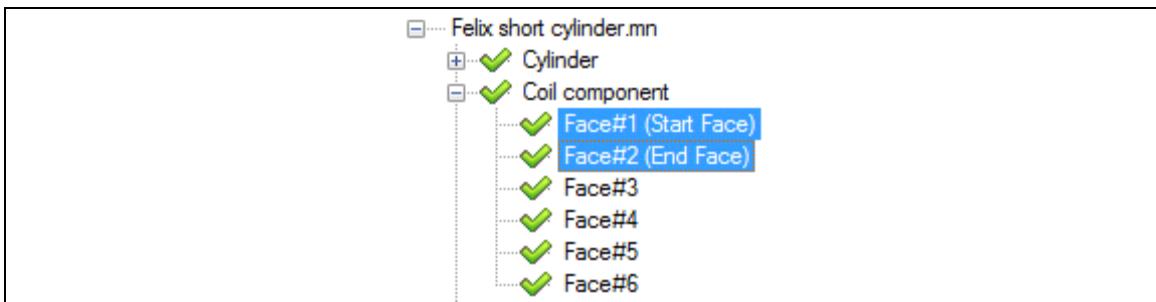
7. On the Boundary Condition toolbar, click  (Field Normal).
The *Field Normal* boundary condition is applied to the selected surfaces.



8. On the File menu, click *Save*.

8 Create the coil

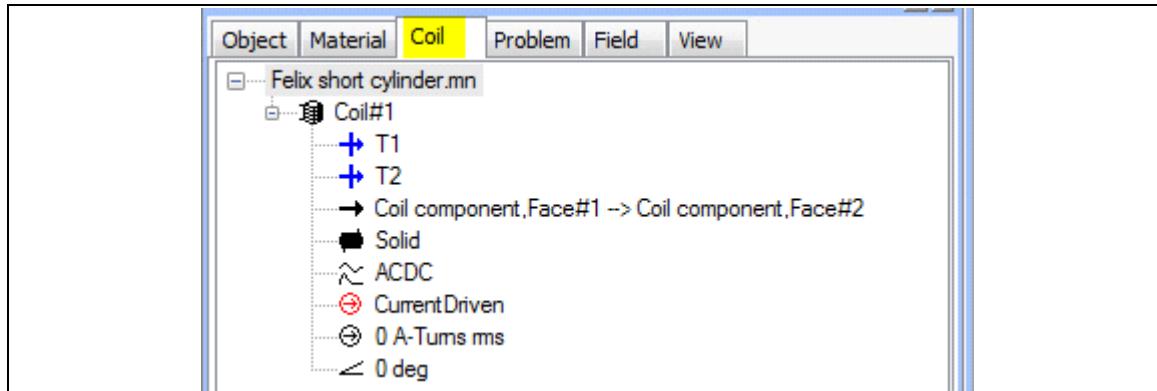
1. On the Object page, select *Face#1(Start Face)* and *Face#2(End Face)* of the *Coil* component.



2. On the Model menu, click *Make Multi-Terminal Coil*.
The coil is listed in the Object page as Coil#1.
3. On the File menu, click *Save*.

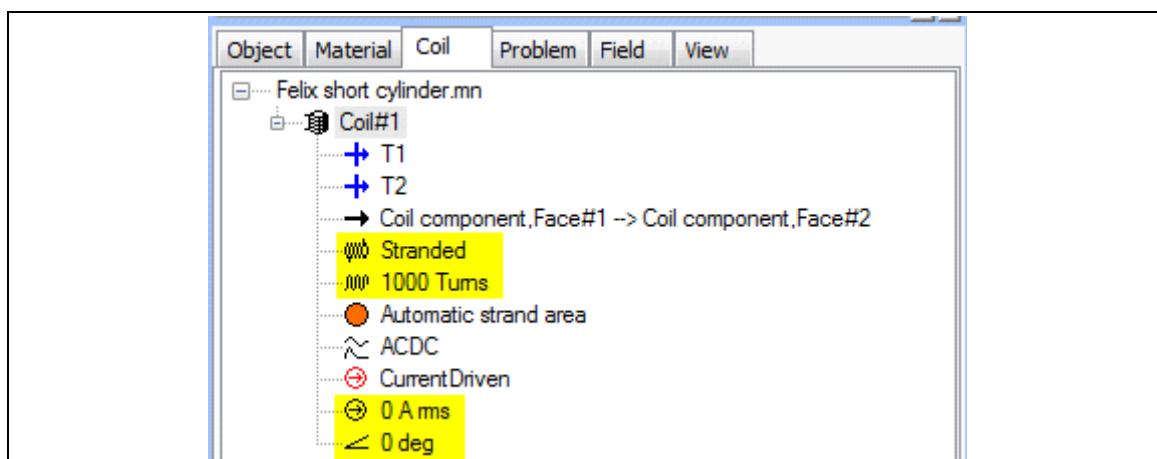
8.1 Edit the coil properties

1. On the Project bar, select the *Coil* tab.
The *Coil* page is displayed.



2. Select the name of the coil (*Coil#1*).
3. On the Edit menu, click *Properties*.
The *Coil* property dialog appears.
4. Do the following:
 - In the *Type* dropdown list, select **Stranded**.
 - In the *Number Of Turns* text box, enter **1000**.
5. Click OK.

The coil page is automatically updated.



6. On the File menu, click *Save*.

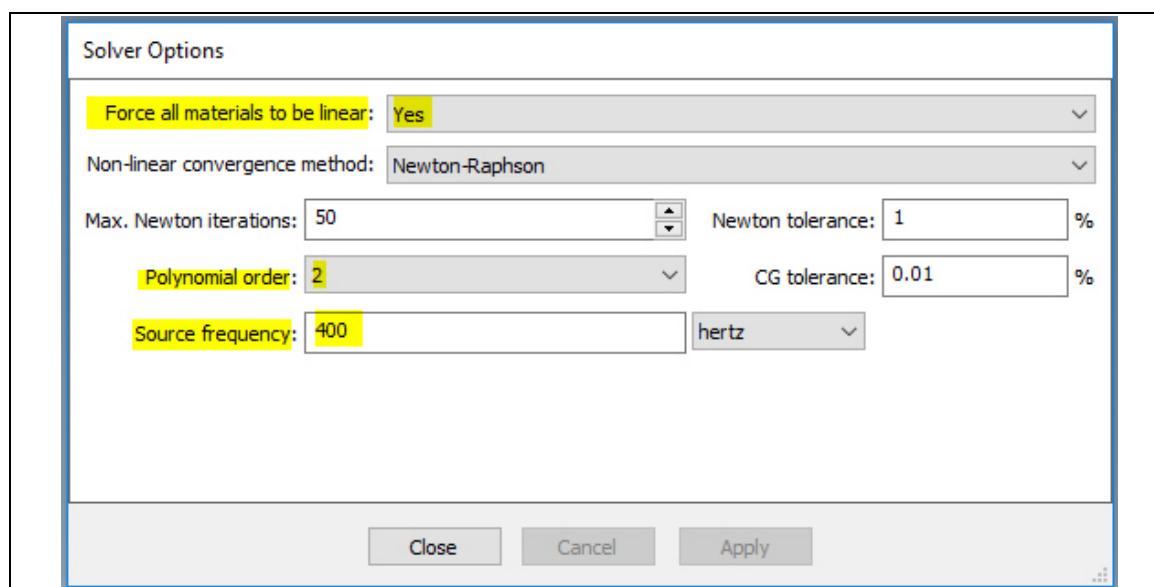
9 Set the source frequency and change the polynomial order

The potential in each element of the mesh is modeled as a polynomial in the spatial coordinates (x, y). In general, higher orders give greater accuracy, but involve greater solution time. For 2D translational models, the default polynomial order is 1. In this tutorial, the order will be changed to 2.

Note Both the source frequency and polynomial order can be set in the Solver Options dialog, or in the Model property page (Solve tab), as well as setting the “Force all materials to be linear” option.

1. On the Solve menu, click *Set Solver Options*.
The *Set Solver Options* dialog appears.
2. Select **Yes** for the “Force all materials to be linear” option.
3. In the *Polynomial order* box, select **2** from the drop-down list.
4. In the *Source Frequency* box, type **400**.

The default unit is Hertz.

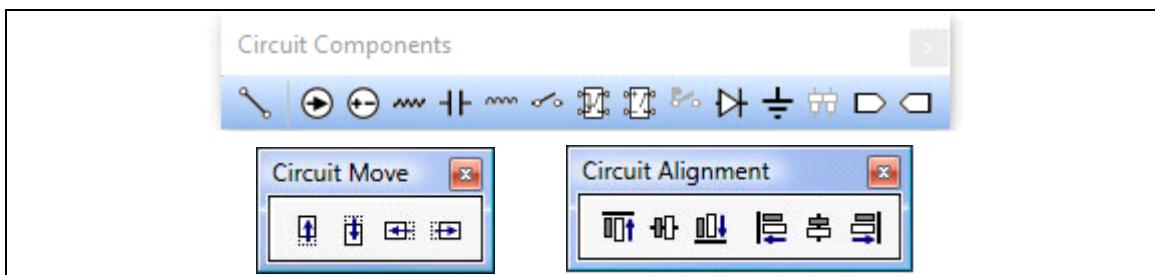


5. Click OK.

10 Create a circuit

The circuit in this model consists of a current source with an exponential waveform and one coil.

For transient analysis, the circuit must be defined in the Circuit window. Circuits are created using the Circuit menu or the Circuit toolbars. In this tutorial, the three Circuit toolbars will be used.

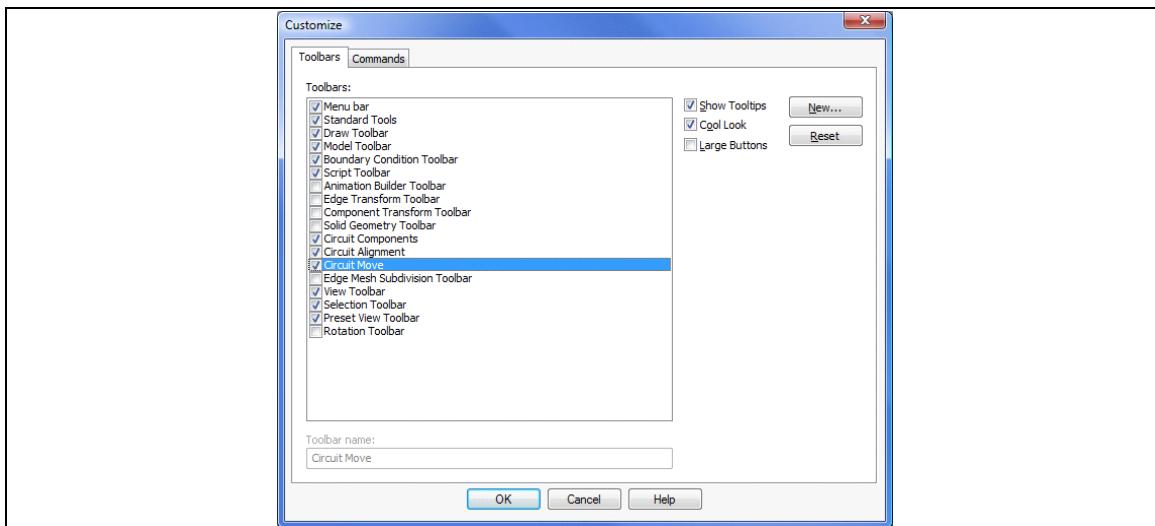


10.1 Display the Circuit toolbars

Note If the circuit toolbars are already displayed, please disregard this procedure.

1. On the Tools menu, click *Customize Toolbars*.

The *Customize* dialog is displayed.



2. From the Toolbars tab, select the following toolbars:

- Circuit Components
- Circuit Alignment
- Circuit Move

Note The toolbars are displayed as they are selected.

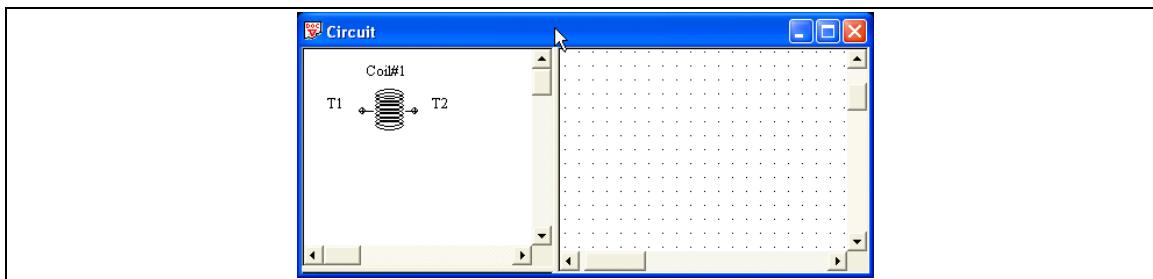
3. Click *OK* to close the Customize dialog and to save your changes.

10.2 Create the circuit

1. On the Window menu, click *New Circuit window*.

A Circuit window opens. The left pane of the Circuit window displays the available coils in the model.

Note The Circuit window in the following images has been minimized.



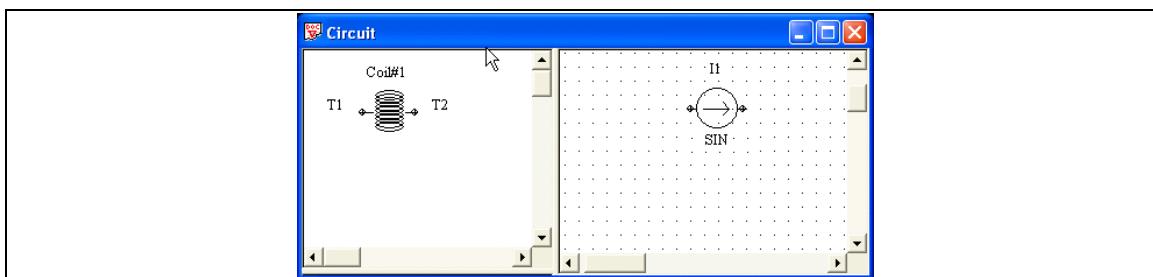
2. On the Circuit Components toolbar, click (Current Source).

If the Circuit Components toolbar is “grayed out”, click the mouse pointer in the right pane of the Circuit window. The right pane must be highlighted before the Circuit toolbars are active.

Tip The Circuit tools are also available on the Circuit menu. For example, to add a current source to the circuit, select Current Source on the Circuit menu.

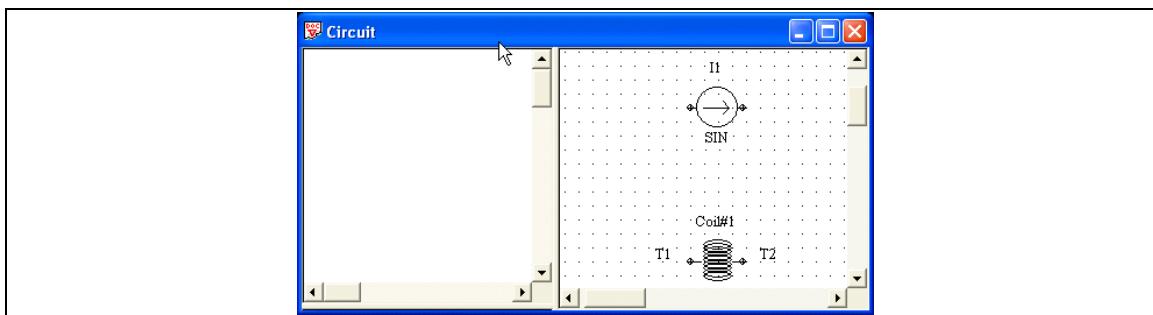
3. Click the mouse pointer in the right pane of the Circuit window.

A current source is added to the window.



4. Select the coil in the left pane of the window, and then drag the coil to the right pane.

If necessary, re-size the window by dragging on an edge of the window.



5. Select the coil and the current source with the mouse pointer.

Tip Press the SHIFT key on your keyboard while selecting the components.

6. On the Circuit Align toolbar, click  (Align Center).

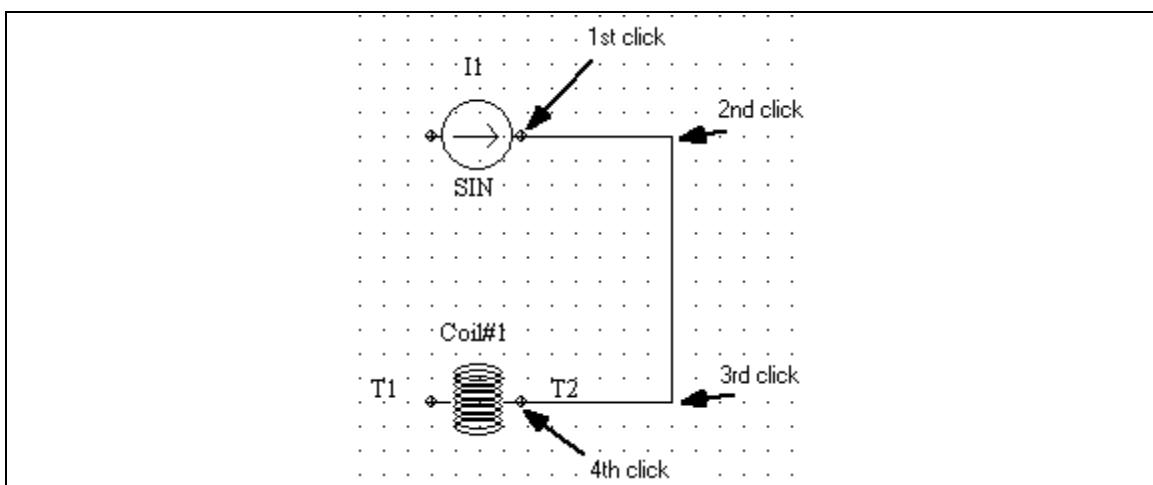
The components are aligned at the position of the last selected component.

7. On the Circuit Components toolbar, click  (Connection).

The Connection tool is used to add connections (wires) between the circuit components.

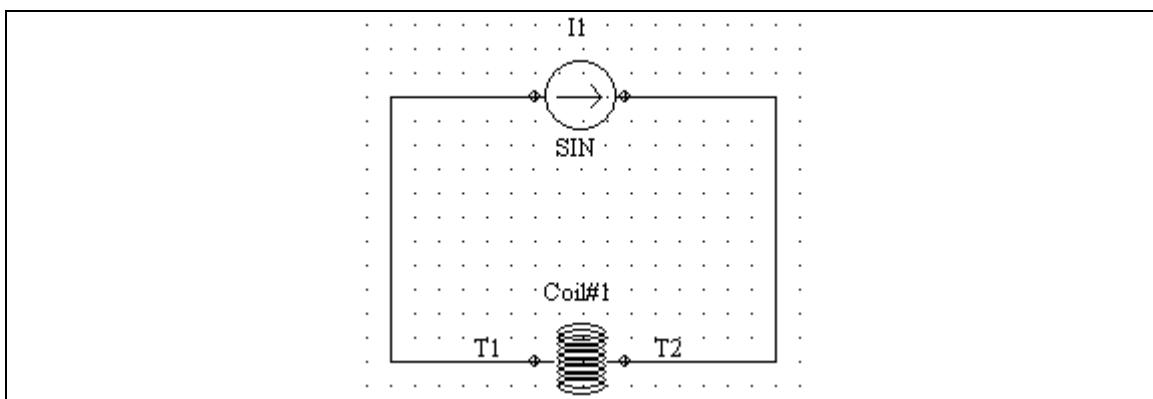
8. Click the mouse pointer on the right terminal of the current source to begin drawing the connection. Continue drawing the connection as shown in the diagram below.

Tip Connections are drawn in the same way as lines are drawn.



9. End the connection on the right terminal of the coil.

10. Draw a second connection as shown in the following diagram.



11. On the File menu, click *Save*.

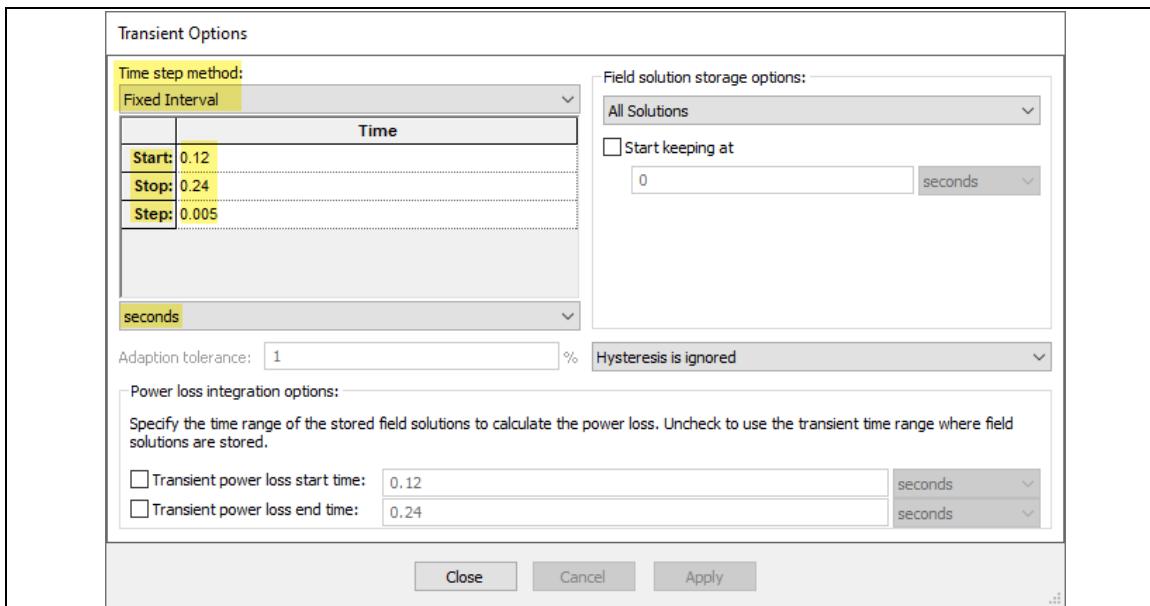
10.3 Set the start, stop and step times

The values of t_1 , T_{d2} , and t_2 are based on the value of the step. To obtain an accurate preview of the waveform, the step value should be set before the waveform properties.

The start, stop, and step times are defined in the Set Transient Options dialog.

1. On the Solve menu, click *Set Transient Options*.

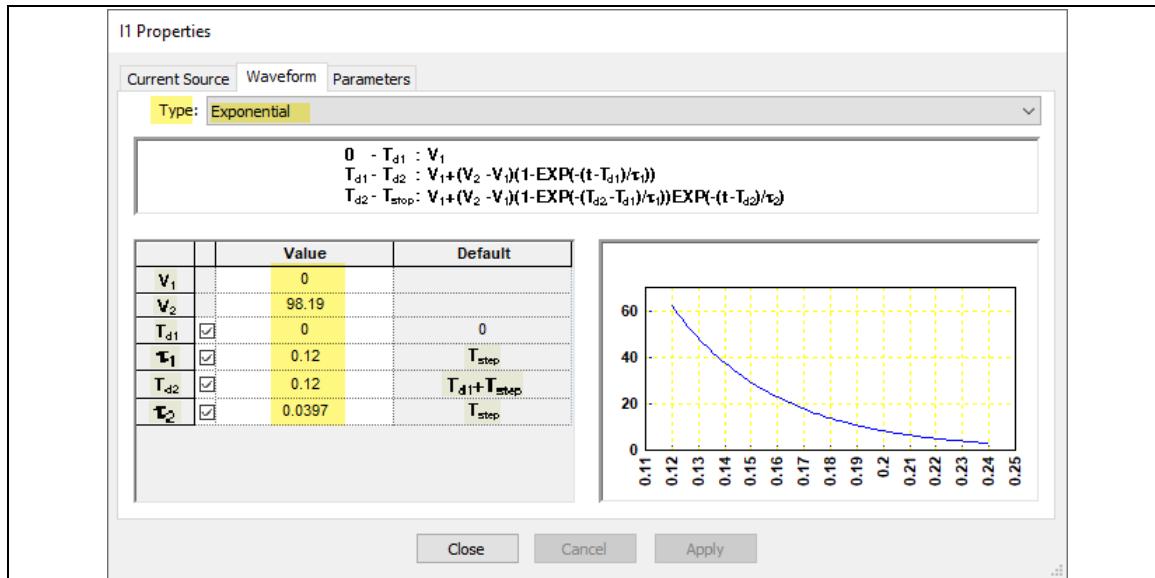
The *Transient Options* dialog appears.



2. Make sure that *Fixed Interval* is selected as the Time Step method and *seconds* as the unit for time, and then make the following modifications for Time:
 - Start = **0.12** Seconds
 - Stop = **0.24** Seconds
 - Step = **0.005** Seconds
3. Click OK.

10.4 Edit the waveform

1. In the right pane of the Circuit window, right-click the current source.
2. On the pop-up menu, click *Properties*.
The *Current Source (I1)* property page appears.
3. Select the Waveform tab.
The *Waveform* property page is displayed.

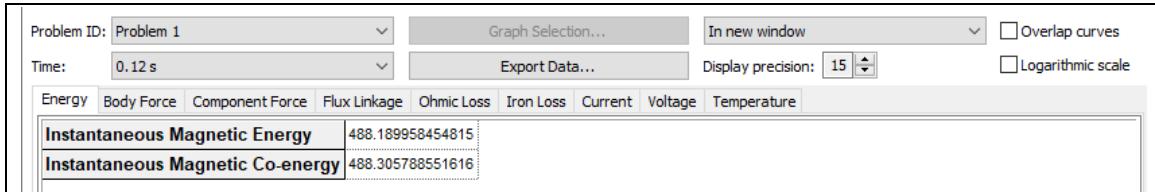


4. In the Type drop-down list, select **Exponential**.
5. Click the **t2** checkbox to enable the fall time constant and all preceding optional values.
6. Insert the following values in the appropriate box:
 - V₁ → **0**
 - V₂ → **98.19**
 - Td1 → **0**
 - t1 → **0.12**.
 - Td2 → **0.12**.
 - t2 → **0.0397**.
7. Click **Apply**.
The waveform is displayed in the property page.
Note Note that the waveform is displayed from the start time of 0.12 seconds.
8. Click **Close**.
9. On the File menu, click *Save*.

11 Solve

- On the Solve menu, click *Transient 3D*.

The *Transient 3D Solver Progress* dialog appears briefly and then the Results window opens.



The Solver Progress dialog automatically exits when the solution is complete.

12 View the solution results

The following results will be reviewed in this section:

- The shaded plot at the first time step
- An animation of the shaded plot across time
- The instantaneous ohmic loss at the last time step of each conducting component
- A graph of the instantaneous ohmic loss across time

Note Before any results can be viewed, you will need to create a new slice and hide the Air box and Cylinder components.

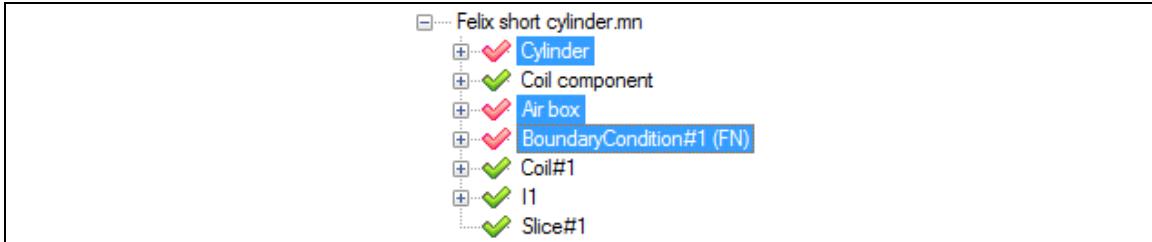
12.1 Create a new slice

1. Before creating a new slice, switch back to the View window by clicking the View tab  located at the bottom of the window.
2. From the Tools menu, click *New Slice*.
3. Select the “Based on the point-normal definition of a plane” option, and enter the following data:
 - Point in the slice: **(0, 0, 0)**
 - Normal of the slice: **(0, 0, 1)**
4. Click OK.

12.2 Hide components

1. From the Object page, select Cylinder, Air box, and Boundary Conditions#1(FN).
2. On the Edit menu, click Toggle Visibility of Selected Components.

The  symbol appearing next to each component object indicates that they are hidden.



12.3 Set the color interpolation and style of the shaded plot

This procedure will set the default for shaded plots to smooth instead of discrete, which is the default.

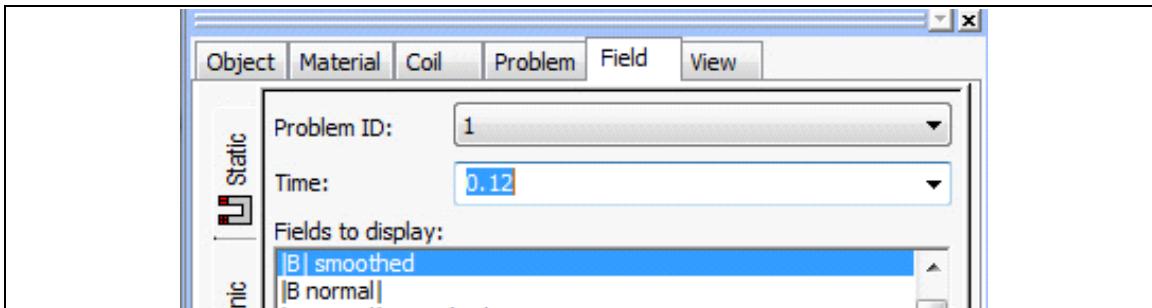
1. On the View menu, click *Default Fields*.
2. On the Project Bar, select the *View* tab.
3. From the View tree, click *Shaded Plot*.
4. On the Edit menu, click *Properties*.
The Shaded Plot Properties page appears.
5. Click the Styles tab and then from the Interpolation drop-down menu, select "Smooth: Displays blended colors between field values".
6. Click OK.

12.4 View the shaded plot

The shaded plot displays shaded lines of the flux density. These shaded lines are the magnitude of the flux density.

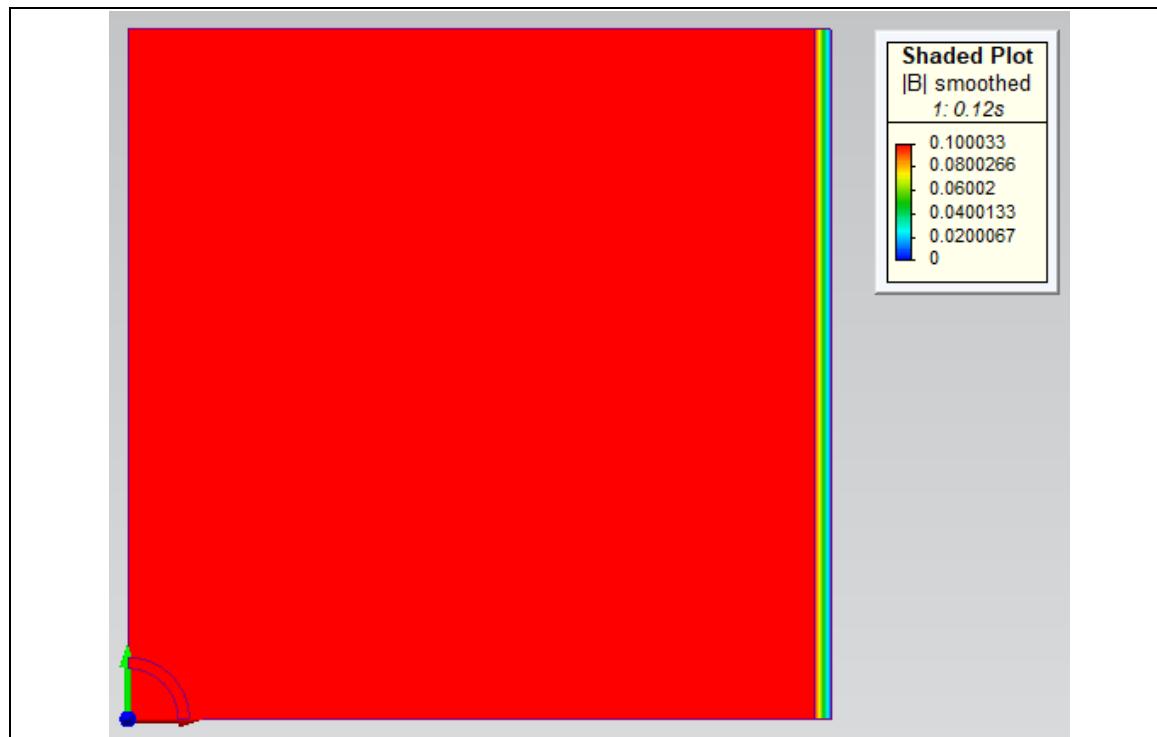
1. On the Project bar, select the *Field* tab.

The *Field* page opens.



2. Select the *Shaded* tab (at the bottom of the Field page).
3. In the *Fields To Display* list, select **|B| smoothed** (should already be selected since it is the default).
4. In the *Time* drop down list, make sure **0.12** is selected (the first time step).

- Tip** To change the default time unit, use the General Model property page (in the Object page, right-click the name of the model, then select Properties). In the Preferred Units for Time drop down list, select the default unit. The unit can be seconds, milliseconds, or microseconds.
5. At the bottom of the Field page, press *Update View*.
The shaded plot is displayed.



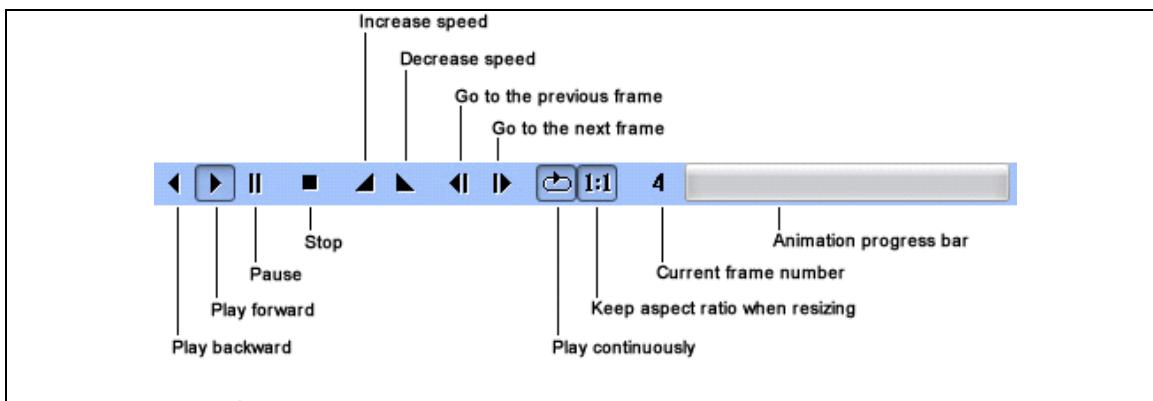
12.5 Animate the shaded plot

An animation is a series of “snapshots”, or frames, that are viewed as a moving image.

1. On the Object page, select *Cylinder*.
2. On the Edit menu, click Toggle Visibility of Selected Components.
The Cylinder component is visible.
3. On the View toolbar, click (Examine Model) and hold down the CTRL key and the left mouse button to form a rectangular box around the cylinder to enlarge the area around it.

4. On the Field page, click  (Animate button).

Please wait a moment while the animation is created. When the animation is complete, an Animation window appears. The animation automatically plays in a loop. Use the *Animation Control* toolbar to control the playing of the animation.



Note The Animation Control toolbar may be hidden from view. If this is the case, maximize the Animation window by double-clicking on its title bar. The Animation Control toolbar should now be visible.

5. Click  (Stop) when you are finished viewing the animation.

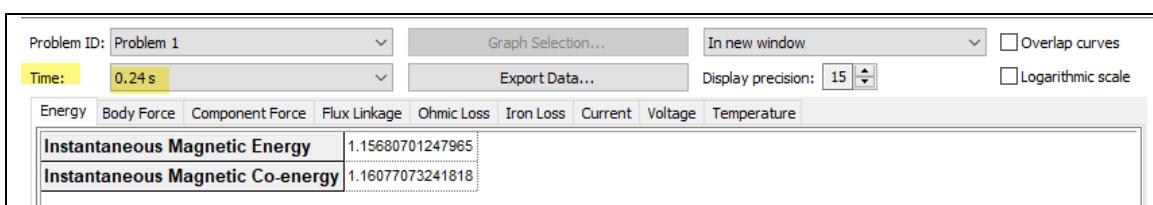
12.6 Save the animation

1. On the File menu, click *Save*.
The *Save As* dialog box appears.
2. In the *File Name* text box, type **Felix short cylinder**.
The animation extension (.ban) is automatically added.
3. In the *Save As* dialog, click *Save*.
The animation is saved.
4. On the File menu, click *Close*.
The Animation window closes.

12.7 View the stored energy

Click the Results tab located at the bottom of the window to display the Results.

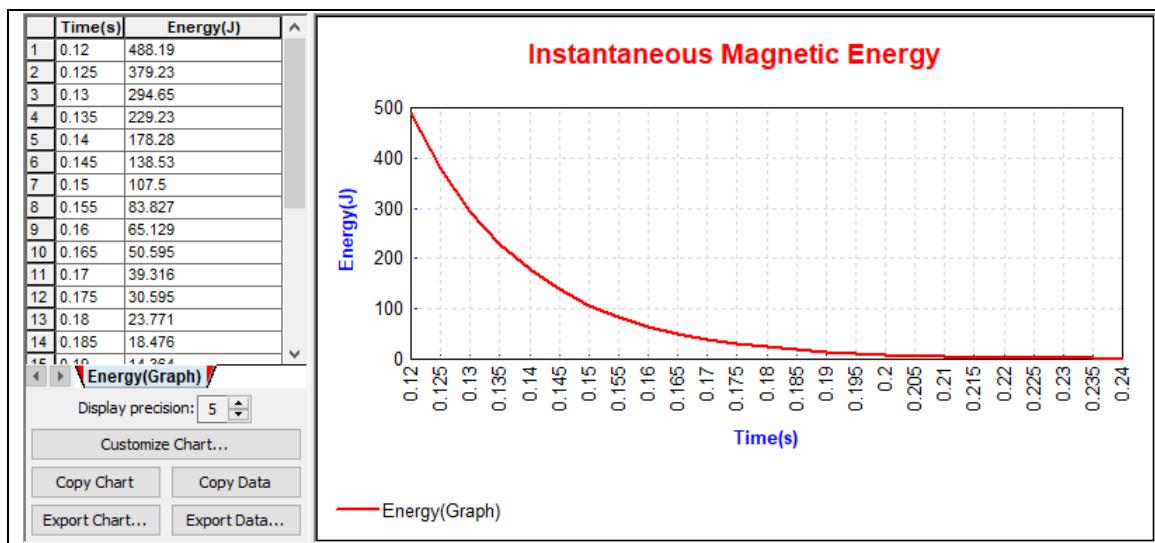
1. Select the *Energy* tab.
The Energy page of the Post Processing bar displays the stored energy in the system.
2. From the *Time* drop down list, select **0.24s**.



12.8 Graph the energy across time

1. Click the mouse pointer in the Instantaneous Magnetic Energy *value* (i.e. 1.15680701247965).
2. Press the *Graph Selection* button.

A new graph window appears. Re-size the window as necessary by dragging on its edges.



12.9 Save the model

You have now completed designing and analyzing the *Felix short cylinder* model.

1. On the File menu, click *Save*.
2. On the File menu, click *Close*.

13 Summary

In this tutorial, you completed the steps in editing the Felix short cylinder for a transient solution. The skills you learned include:

- Creating a new material
- Creating a circuit
- Defining an exponential waveform
- Viewing a shaded plot
- Animating a shaded plot
- Viewing the stored energy in the system
- Graphing the energy across time.

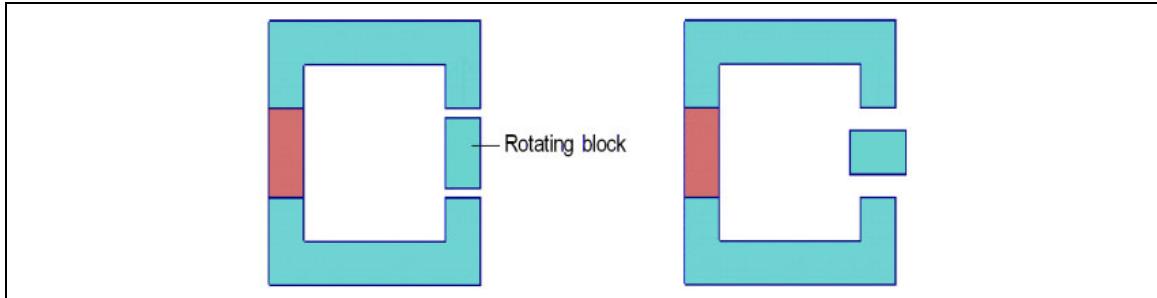
Tutorial #10

3D Parameterization

C-core with a rotating block

1 Modeling plan

This tutorial builds and solves a C-core with a rotating block. Using Simcenter MAGNET's parameterization feature, the rotating block is solved at eight positions: 0, 45, 90, 135, 180, 225, 270, and 315 degrees.



The geometry of the C-core and the permanent magnet is drawn using the mouse pointer. The geometry is then swept into 3D components and filled with a material. The rotating block is parameterized and eight models are created.

After solving the model, the shaded plot is viewed on a slice through the model. The shaded plot is then animated across the solutions.

2 Open a new model

1. Start Simcenter MAGNET.
The Main window appears.
2. If Simcenter MAGNET is already running, select *New* from the File menu to open a new model.
If you have already used Simcenter MAGNET, the window displays the settings that were last active.
Click  on the top right corner of the Simcenter MAGNET Main window to maximize the window.

2.1 Name the model

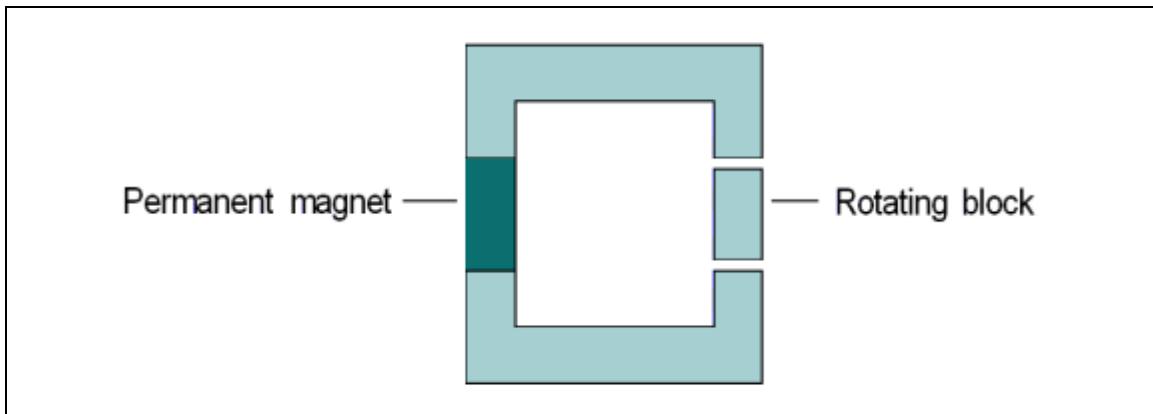
1. On the File menu, click *Save As*.
2. In the *Save As* dialog box, enter **C-core with a rotating block** as the name of the model.
3. Choose the drive and directory where you want to place the model.
4. Click *Save As*.

2.2 Set the maximum element size

1. In the Object page of the Project bar, select the model (i.e. *C-core with a rotating block.mn*).
2. On the Edit menu, click *Properties*.
The Properties dialog appears.
3. Select the *Mesh* tab.
4. Click inside the *Maximum element size* checkbox, and then type **0.6** in the text box.
5. Click *OK*.

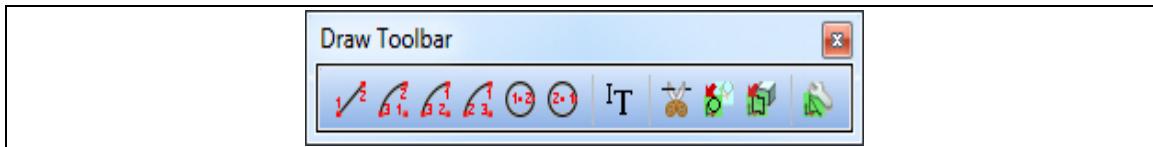
3 Build the geometric model

The C-core with a rotating block is built from four components. Two components form the C-core. The permanent magnet and the rotating block form the third and fourth component.



4 Draw the geometry of the C-core

Drawing tools are located on the Draw toolbar and the Draw menu. Edges are drawn by using the mouse pointer or by entering coordinates through the keyboard. In this tutorial, edges are drawn using the mouse pointer.



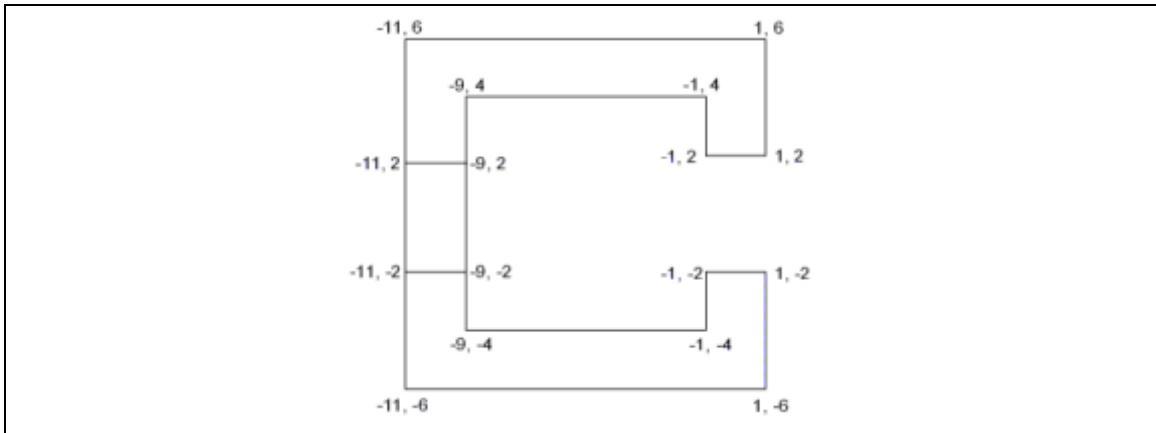
4.1 Set the drawing space

The geometry is drawn using the Snap to Grid drawing aid. When *Snap to Grid* is selected, the points drawn by the mouse pointer are snapped, or pulled, to the construction grid points.

1. Make sure the Construction Grid is displayed.
If the grid is not visible, select *Construction Grid* on the View menu.
2. On the View menu, click *Set Construction Grid*.
The Set Construction Grid dialog appears.
3. On the *Set Construction Grid* dialog, enter the following:
 - Minimum X and Y text boxes: **-20**.
 - Maximum X and Y text boxes: **20**.
4. Click OK on the *Set Construction Grid* dialog.
5. On the Draw menu, make sure that *SnapMode* is set to *Grid*.

4.2 Draw the geometry

The geometry of the C-core is shown in the diagram below.

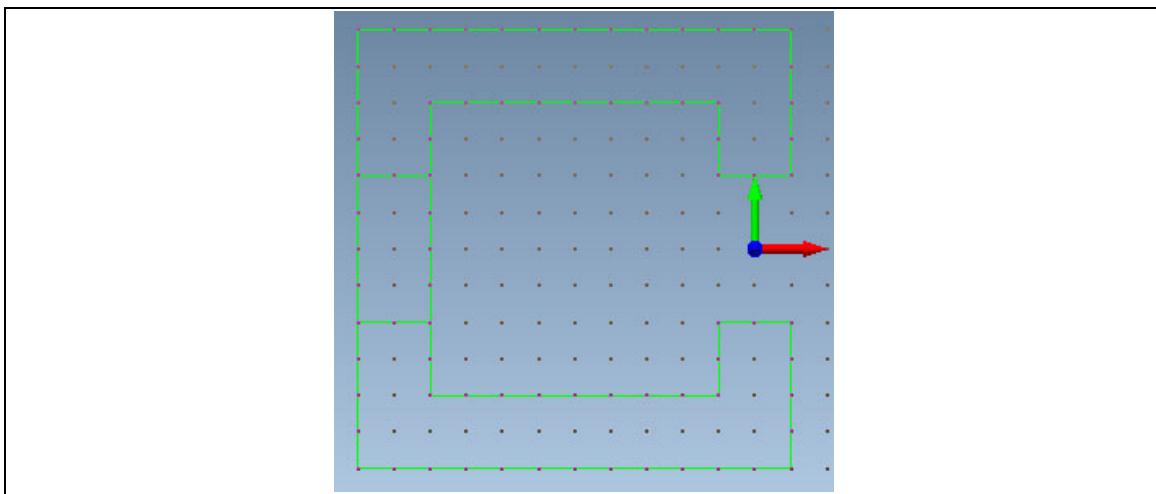


1. On the Draw toolbar, click (Line drawing tool).
2. Using the mouse pointer, place the coordinates using the above diagram and the status bar (shown below) as guides.



Note To stop drawing, press ESC.

When you are finished drawing the C-core, the geometry should look like the following diagram.

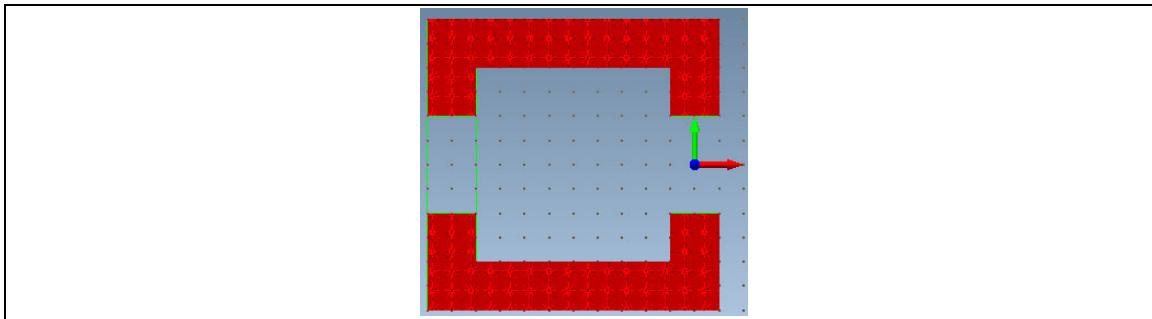


4.3 Make the components of the C-core

Three-dimensional components can now be made from the surfaces of the C-core.

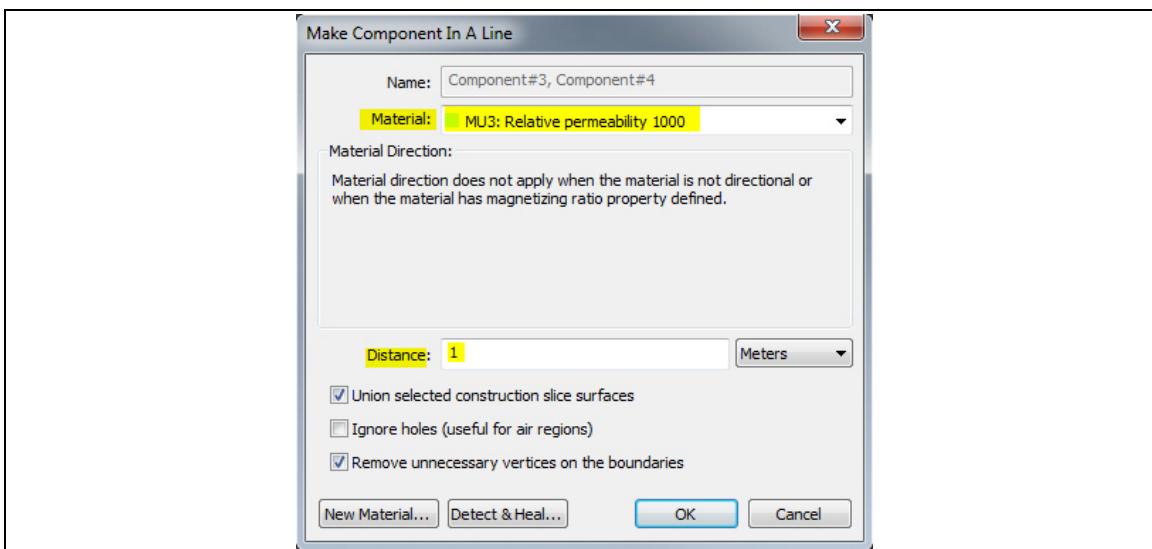
1. On the Edit menu, click Select Construction Slice Surfaces.
2. Hold down the Shift key and click the mouse pointer inside the top and bottom surfaces, respectively, of the C-core.

The surfaces are highlighted when selected.



3. On the Model toolbar, click  (Make Component in a Line tool).

The *Make Component In A Line* dialog box appears.



4. In the *Material* drop down list, select **MU3: Relative permeability 1000**.
5. In the *Distance* box, enter **1**.
6. Click **OK** to accept the settings.
7. On the File menu, click *Save*.
The components are created.
8. Rename the components, using the Properties page (i.e. General tab/Name):
 - Component#1 to **Core upper half**
 - Component#2 to **Core lower half**

4.4 Make the component of the permanent magnet

1. Click the mouse pointer inside the construction surface of the permanent magnet.
2. On the Model toolbar, click  (Make Component in a Line tool).
3. In the **Name** box, enter **Permanent Magnet**.
4. In the *Material* drop down list, select **PM10: Br 1.0 mur 1.0**.
5. In the *Material Direction Type* drop down list, select **Uniform** (default).
6. In the *Direction* text box, enter **(0, 1, 0)** (default).
7. In the *Distance* box, enter **1**.
8. Click **OK** to accept the settings.
9. Click **Save**.

5 Create the rotating block

5.1 Draw the geometry

The geometry of the rotating block is now added to the C-core.

1. If the Keyboard Input bar is not already displayed at the bottom of the Main window, click *Keyboard Input Bar* on the Tools menu.
2. Make sure that  (Cartesian) and  (Absolute) are selected on the Keyboard Input bar.
3. On the Draw toolbar, click  (Line drawing tool).
4. In the Keyboard Input bar, enter the following coordinates for the polyline.

Start coordinates -1, 1.6 Press ENTER

End coordinates -1, -1.6 Press ENTER

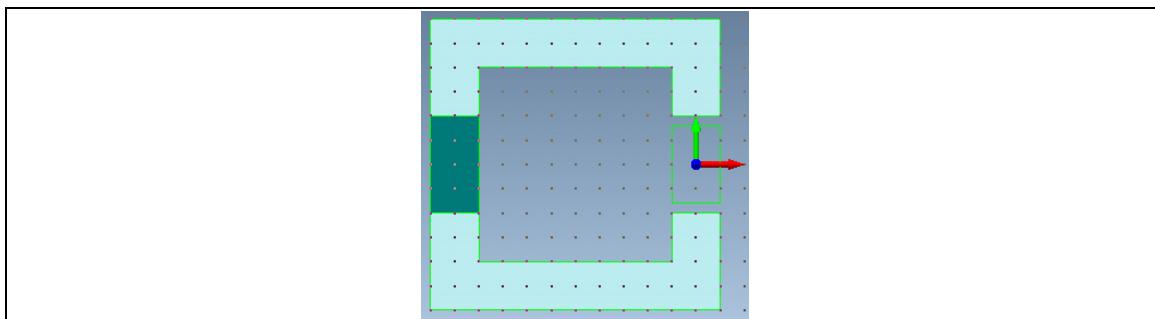
End coordinates 1, -1.6 Press ENTER

End coordinates 1, 1.6 Press ENTER

End coordinates -1, 1.6 Press ENTER

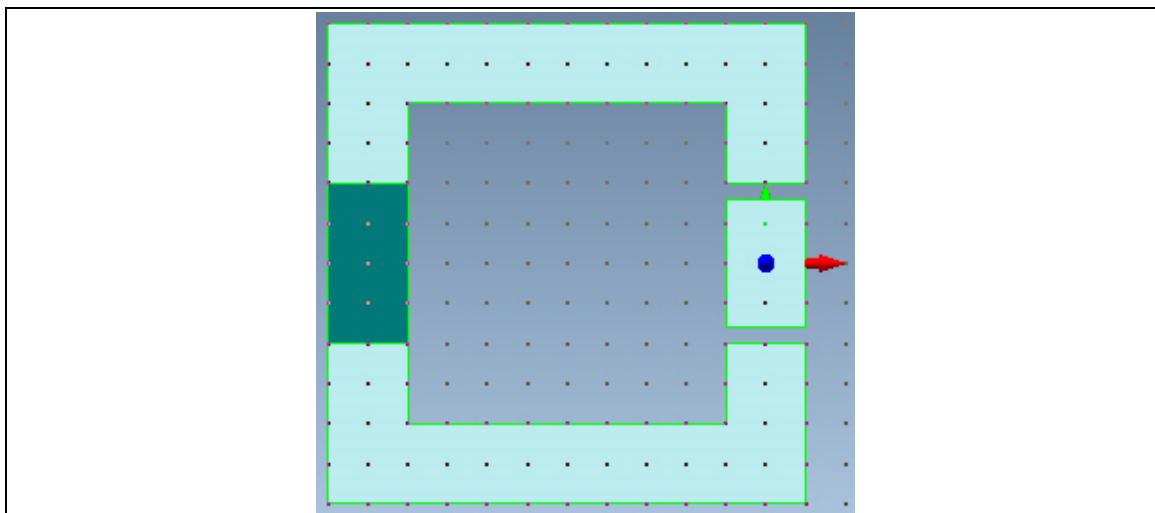
5. Press ESC.

The model should look like the diagram below.



5.2 Make the component of the rotating block

1. On the Edit menu, click Select Construction Slice Surfaces.
2. Click the mouse pointer in the surface of the rotating block.
3. On the Model toolbar, click  (Make Component in a Line tool).
The *Make Component in a Line* dialog box appears.
4. In the *Name* text box, enter **Rotor**.
5. In the *Material* drop down list, select **MU3: Relative permeability 1000** (now located under Model Materials since it has already been assigned previously in this model).
6. In the *Distance* box, enter **1**.
7. Click OK to accept the settings.
The component is created.



8. Click Save.

6 Add a boundary condition by creating an air box

Each model needs a definition of where its boundary in space exists and how the magnetic flux will flow in relation to this boundary.

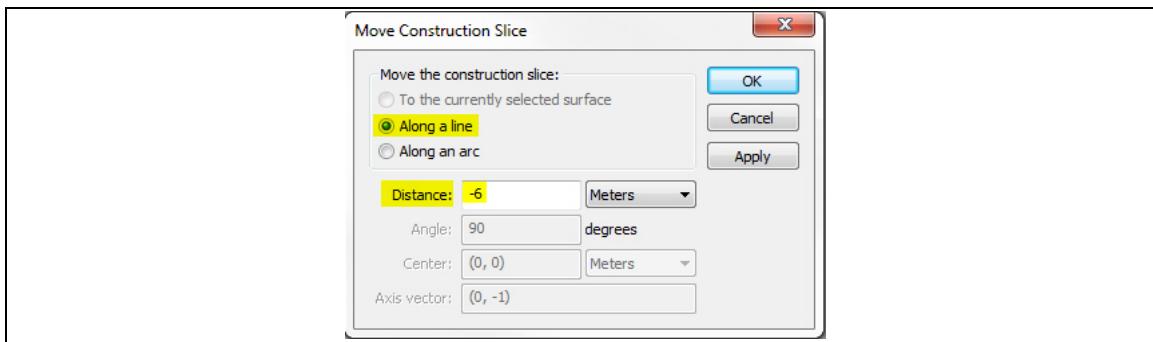
In this model, a tangential flux boundary will be added around the C-core. This boundary forces the magnetic flux lines to flow tangential to the boundary.

The boundary is created by sweeping an air box around the C-core. The default boundary condition, tangential flux, is automatically applied to the outermost surfaces of the air box.

6.1 Move the construction slice

The air box begins below the starting surfaces of the C-core.

1. On the Draw toolbar, click  (Move Construction Slice tool).
The Move Construction Slice dialog appears.



2. Make sure *Along A Line* is selected.
3. In the *Distance* box, enter **-6**.
4. Click OK.

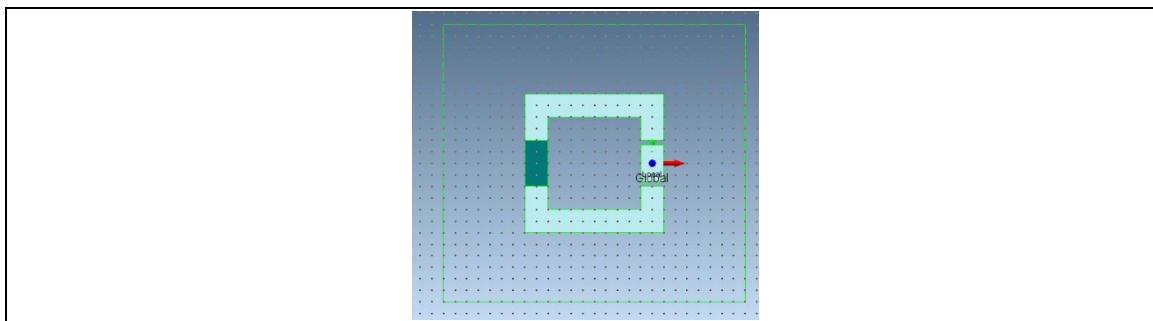
6.2 Draw the geometry

1. On the Draw toolbar, click  (Line drawing tool).
2. On the View menu, click *Update Automatically*.
3. In the Keyboard Input bar, enter the following coordinates:

Start coordinates	-18, 12	Press ENTER
End coordinates	8, 12	Press ENTER
End coordinates	8, -12	Press ENTER
End coordinates	-18, -12	Press ENTER
End coordinates	-18, 12	Press ENTER

4. Press ESC.

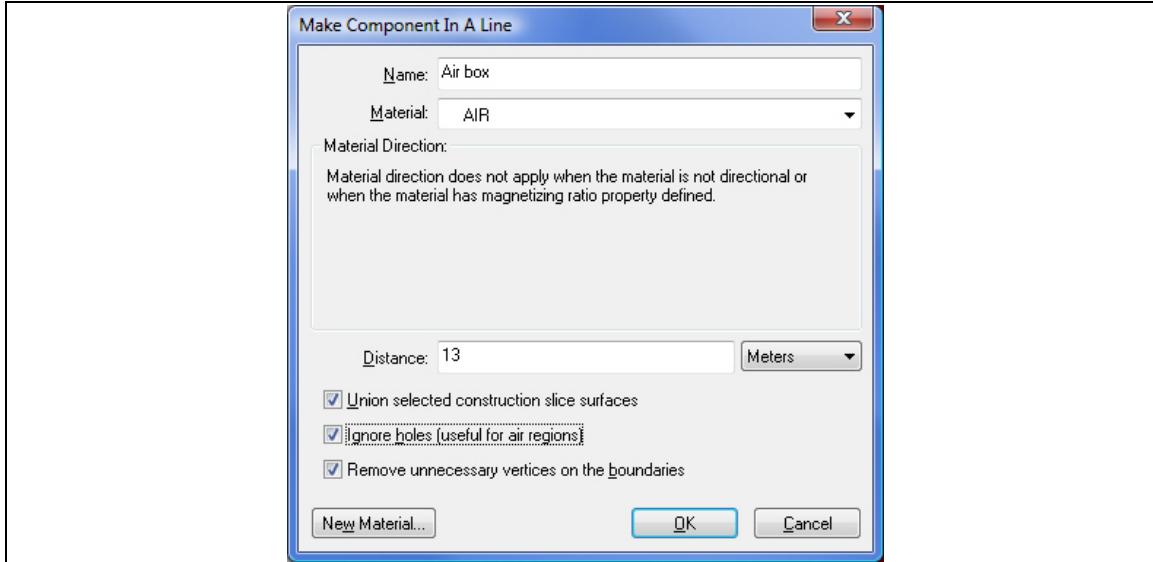
The model should look like the diagram below, with the C-core centered in the air box.



6.3 Make the component

1. On the Edit menu, click Select Construction Slice Surfaces.
2. Click in the surface of the air box.
3. On the Model toolbar, click  (Make Component in a Line tool).

The Make Component in a Line dialog appears.



4. In the *Name* box, enter **Air box**.
5. In the *Material* drop down list, make sure **AIR** is selected.
6. In the *Distance* box, enter **13**.
7. Select the “Ignore holes (useful for air regions)” checkbox.
This feature eliminates the need to delete the construction slice edges before extruding the *Air box*.
8. Click **OK**.
9. Click **Save**.

7 Parameterize the rotating block

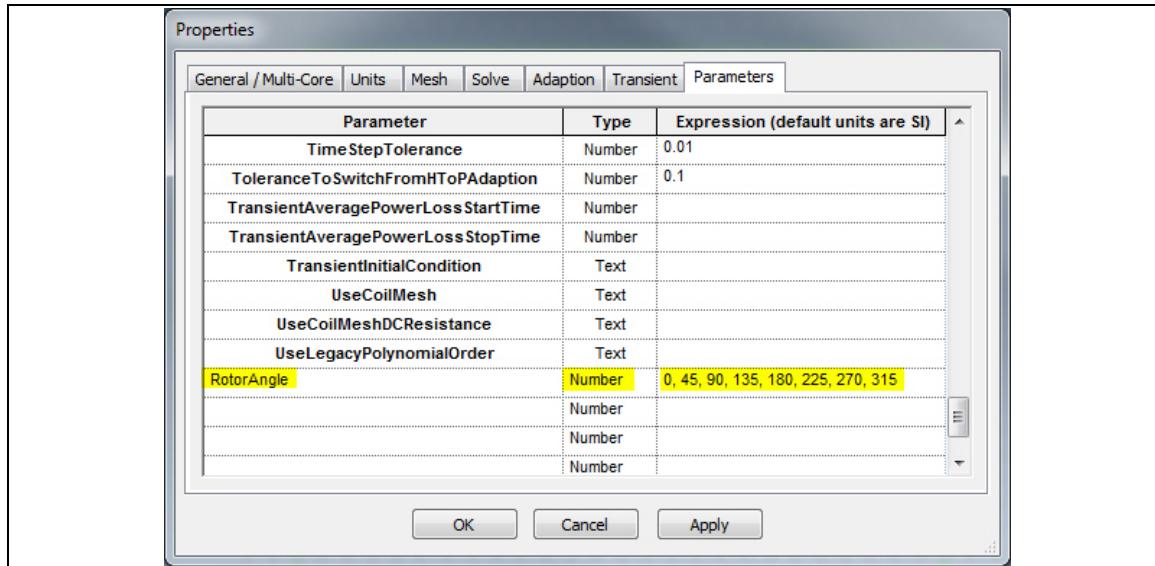
A user-defined parameter will be created to rotate the block at eight positions: 0, 45, 90, 135, 180, 225, 270, and 315 degrees. User-defined parameters are created in the Model parameters page and are available for use throughout the model.

The user-defined parameter will then be used as an expression in the Component property page.

7.1 Create a user-defined parameter

1. In the Object page, click the model (i.e. *C-core with a rotating block.mn*).
2. On the Edit menu, click *Properties*.
The Properties dialog appears.
3. Select the *Parameters* tab.

The model's Parameters page appears.



4. In the *Parameters page*, use the scroll bar to display an empty line.
5. In the *Parameter* column of the empty line, enter **RotorAngle**.



Note Make sure you do not add any spaces before, between, or after the words.

6. Click the mouse pointer in the *Type* column.
A drop down list appears.
7. Choose *Number* from the drop down list.
8. In the *Expression* column, type the following: **0, 45, 90, 135, 180, 225, 270, 315**.
9. Click *Apply*.

Clicking *Apply* leaves the dialog box open for the next procedure, which is to use the *RotorAngle* parameter as an expression in the Rotor component's property.

8 Apply the user-defined parameter

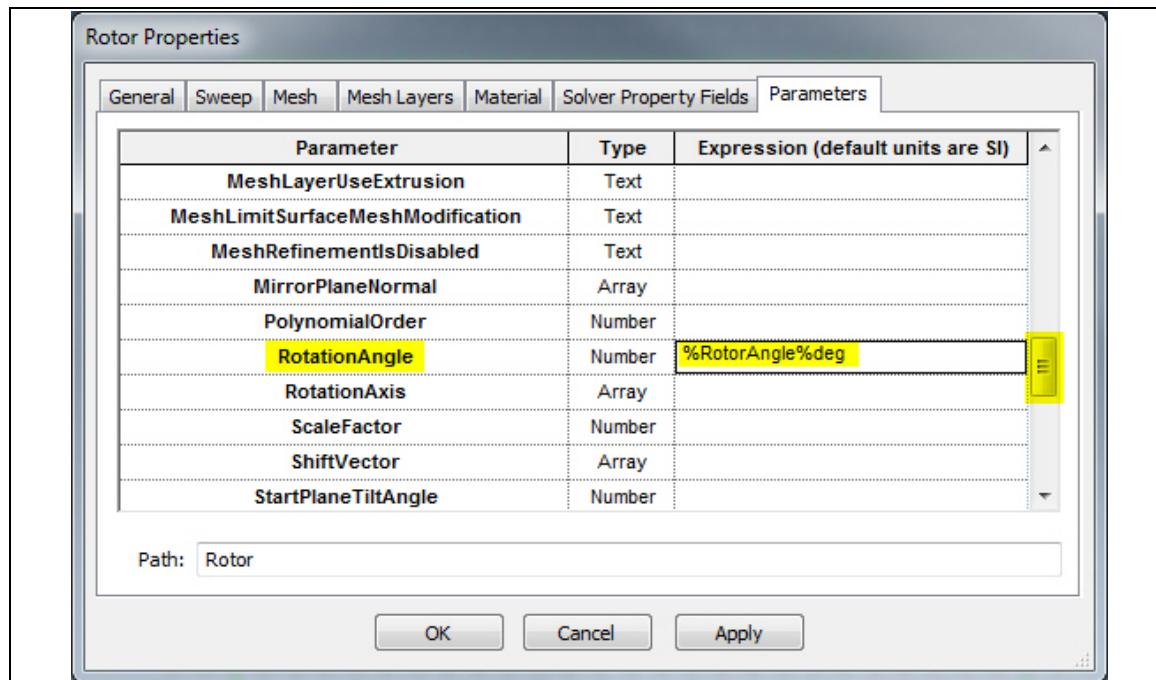
The position of the rotor is parameterized in the Rotor component's property page.

8.1 Open the Component property page

- With the Parameters page still open from the previous procedure, select *Rotor* on the Object page.

Note The Rotor Properties dialog now has focus.

- Use the scroll bar on the right side of the page to scroll down to the *RotationAngle* parameter.



- In the *Expression* column of the *RotationAngle* parameter, enter the following:
%RotorAngle%deg

Note The user-defined parameter name must be prefaced by the “%” character. The default rotation unit is radians. The expression “%deg” specifies degrees as the rotation unit.

- Press TAB.
- Click OK.

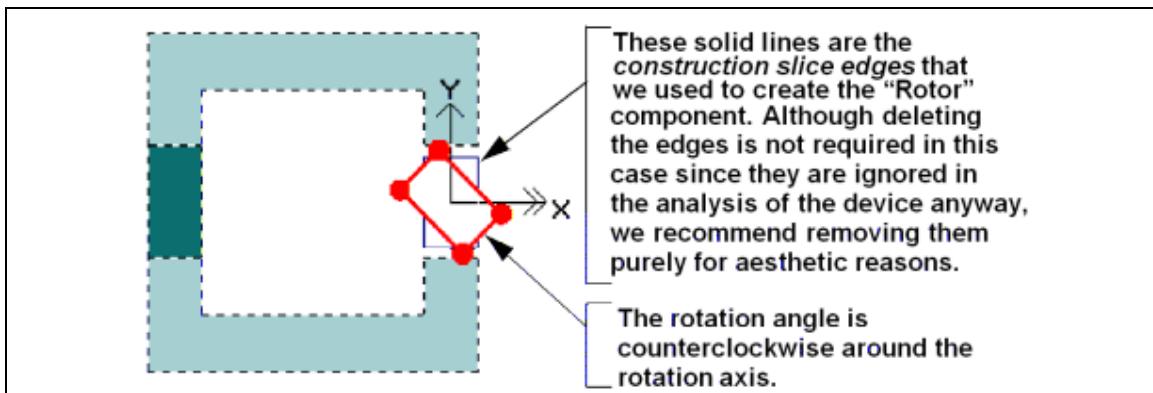
9 View the instantiated models

The instantiated models are viewed in the Problem page of the Project bar.

1. On the Project bar, select the *Problem* tab.
The *Problem page* appears. (The angle is expressed in radians in the Problem page.)
2. Click **2** (Problem 2 tab) in the second row of the Problem page.
The row is highlighted.

Object	Material	Coil
Problem	Field	View
1	<input checked="" type="checkbox"/>	0
2	<input checked="" type="checkbox"/>	45
3	<input checked="" type="checkbox"/>	90
4	<input checked="" type="checkbox"/>	135
5	<input checked="" type="checkbox"/>	180
6	<input checked="" type="checkbox"/>	225
7	<input checked="" type="checkbox"/>	270
8	<input checked="" type="checkbox"/>	315

3. In the Problem page, click **Update View** (Update View button).
The View window is updated to display Problem 2.



4. In the Problem page, click the *Problem 1 tab*, and then click the *Update View* button
The View window is updated to redisplay Problem 1.

9.1 Delete Construction Slice edges

Since the edges on the construction slice remain even after being swept into a component, we must remove them before proceeding to the next step of the tutorial.

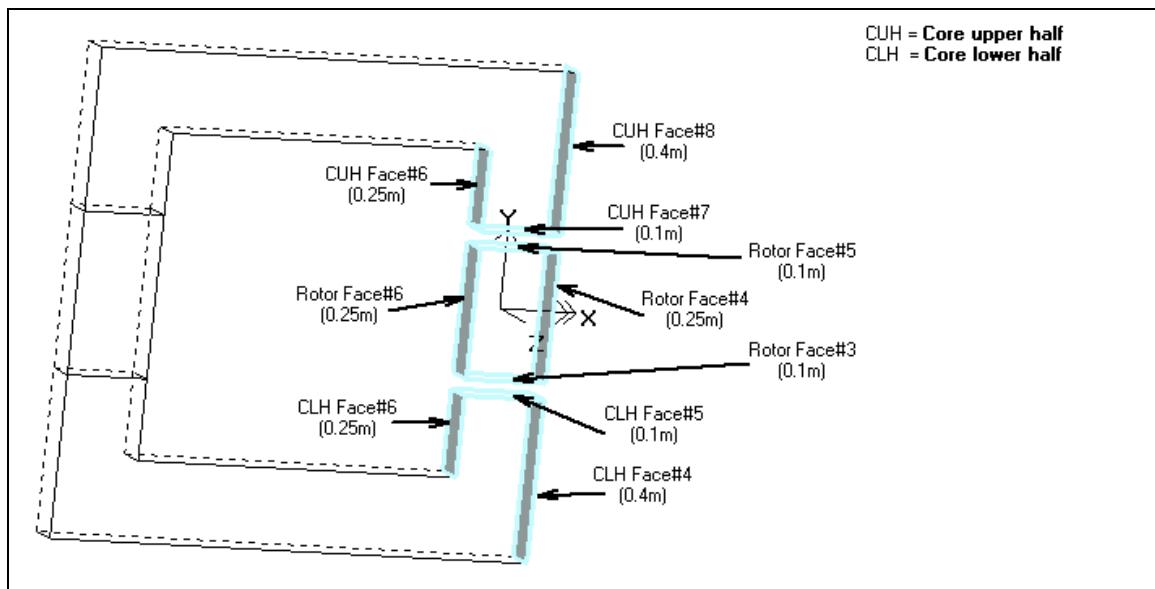
1. On the Edit menu, click Select Construction Slice Edges.
2. On the Edit menu, click *Select All*.
3. On the *Standard Tools* toolbar, click (Delete).

10 Modify the mesh

In the 3D finite element method of analysis, the model is divided into a mesh of tetrahedral-shaped elements. Each element is defined by four vertices (nodes). The vector field inside each element is represented by a polynomial with unknown coefficients. The finite element analysis is the solution of the set of equations for the unknown coefficients.

The accuracy of the solution depends upon the nature of the field and the size of the mesh elements. In regions where the direction or magnitude of the field is changing rapidly, high accuracy requires small elements. One method of increasing mesh density is to set the *maximum element size* for a component volume or specific faces of a component. The following procedure will demonstrate this method.

Note *Maximum element size* modifications (in parentheses) will be applied to each component face shown in the diagram below.

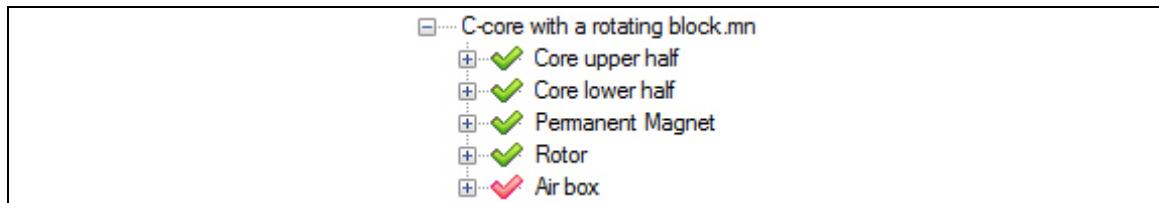


10.1 View the initial mesh

Before changing the *maximum element size*, the default initial mesh can be viewed. For this exercise, the Air box component will be hidden from the display so that the C-core is more easily viewed.

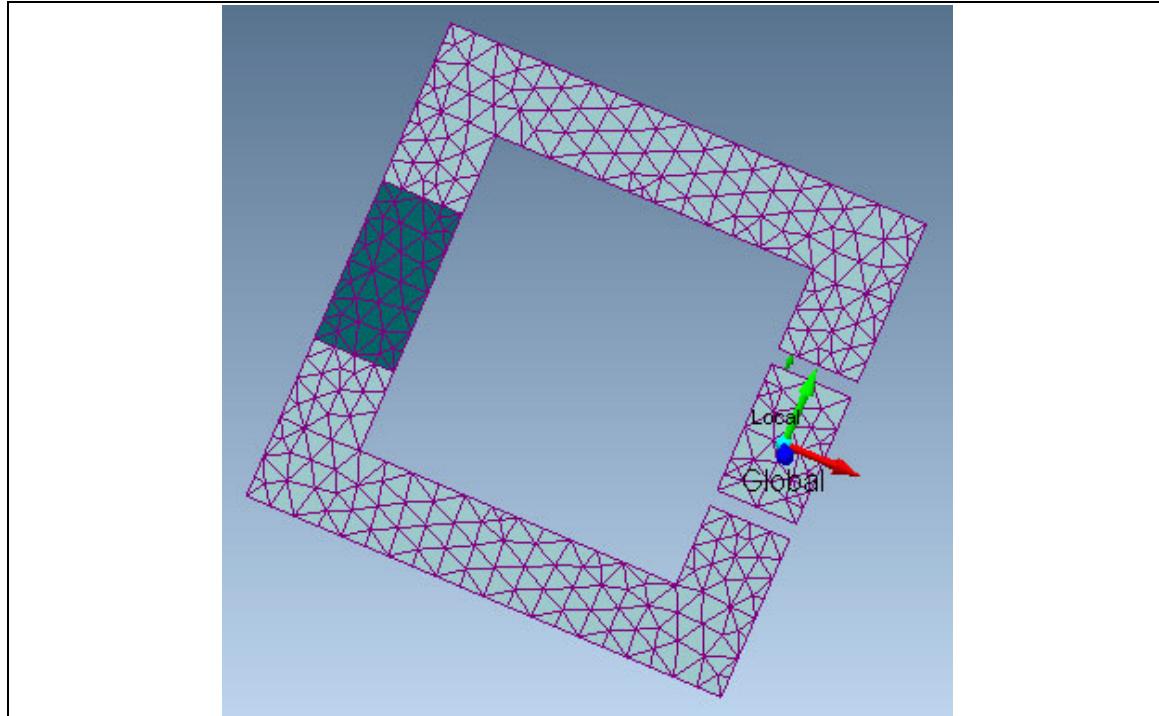
1. In the Object page, select *Air box*.
2. Click the right mouse button.
A pop-up menu appears.

3. On the pop-up menu, toggle *Visible* so that the check mark disappears.
On the Object page, a symbol is displayed next to the *Air box* component.



4. On the View toolbar, click (Automatic View All).
5. On the View toolbar, click (Examine Model).
6. Holding down the left mouse button, drag the mouse pointer in one or more of the following directions:
 - Drag down to rotate the display downward.
 - Drag up to rotate the display upward.
 - Drag left to rotate the display toward the left.
 - Drag right to rotate the display toward the right.
7. Release the mouse button once the display is rotated so that it looks similar to the illustration below. The model is displayed as a wireframe during the rotation.
8. On the View menu, click *Initial 3D Mesh*.

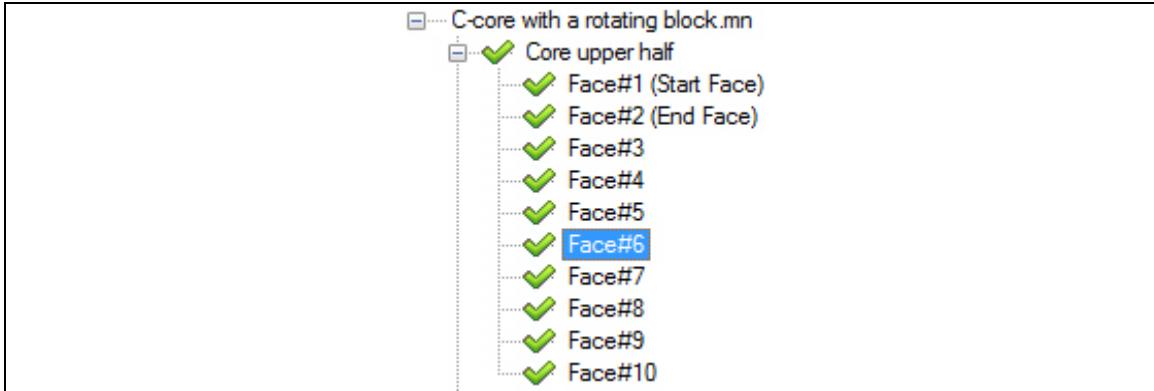
The initial mesh appears in the View window.



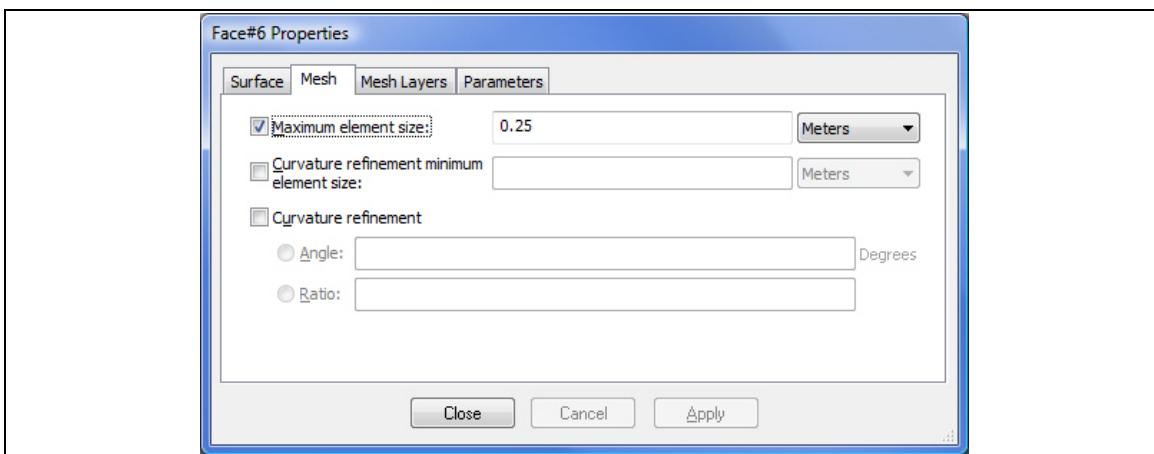
9. On the View menu, click *Solid Model*.

10.2 Set the maximum element size for each component

1. Place the cursor over the *Object* tab of the Project bar, right-click and then select *Expand All*.
The Object tree is expanded to show the numerous faces that make up each component.
2. In the Object page, select *Face#6* of the *Core upper half* component.



3. On the Edit menu, click *Properties*.
The Properties dialog appears.
4. Select the *Mesh* tab.
5. Click inside the *Maximum element size* checkbox, and then type **0.25** in the text box.



6. Click *Apply*.

Tip Clicking *Apply*, instead of *OK*, keeps the dialog open and allows us to proceed to the next component faces without having to repeat steps 3 and 4.

7. In the Object page, select *Face#7* of the *Core upper half* component.
Notice that the text in the Properties dialog Title Bar has now changed to read *Face#7 Properties*.
8. Click inside the *Maximum element size* checkbox, and then type **0.1** in the text box.
9. Click *Apply*.

10. Repeat steps 7 through 9 for each of the remaining component faces, assigning the following values:

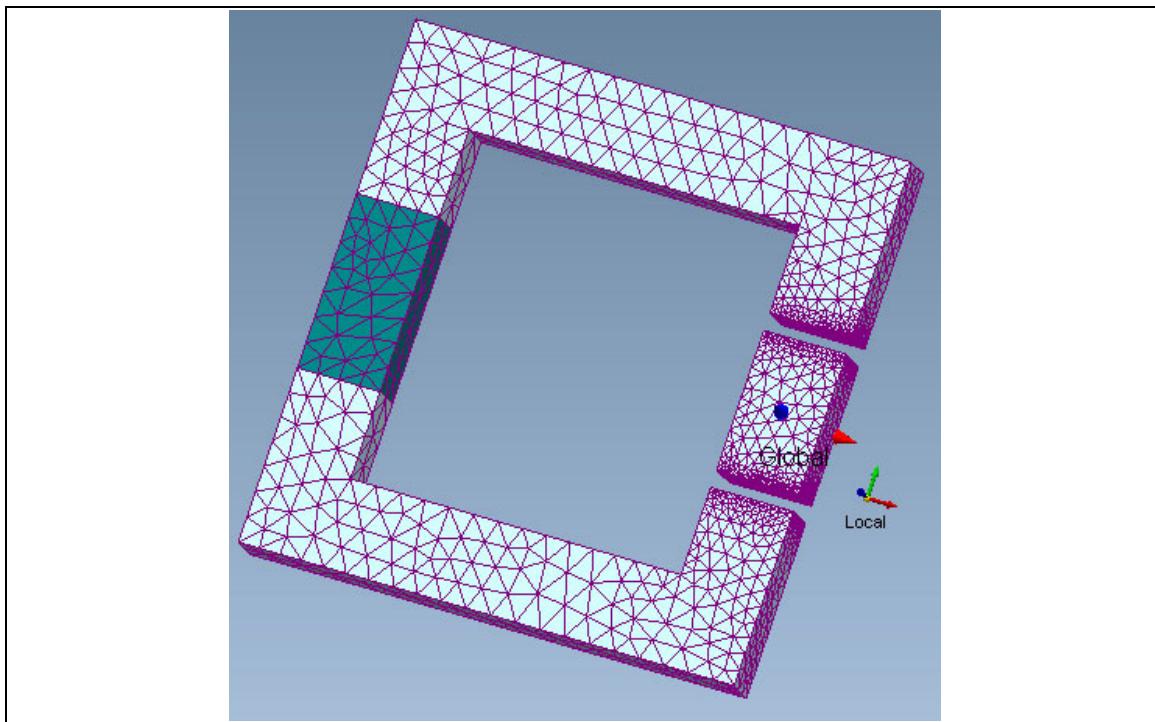
Component face	Maximum element size
Core upper half Face#8	0.4m
Core lower half Face#4	0.4m
Core lower half Face#5	0.1m
Core lower half Face#6	0.25m
Rotor Face#3	0.1m
Rotor Face#4	0.25m
Rotor Face#5	0.1m
Rotor Face#6	0.25m

11. Once all the *Maximum element size* modifications have been done, click OK.

10.3 View the changes to the mesh

1. On the View menu, click *Initial 3D Mesh*.

The modified initial mesh appears in the View window and should look like the following illustration.



2. On the View menu, click *Solid Model*.

10.4 Collapse the Object tree and make the Air box component visible

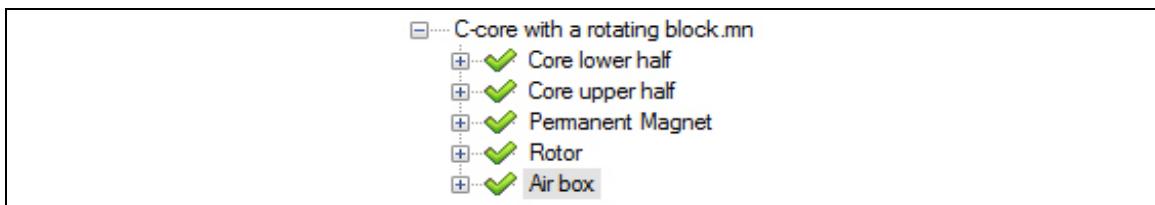
Now that the model is ready to be solved, we will make the *Air box component* visible again.

1. Place the cursor over the *Object* tab of the Project bar, right-click and then select *Collapse All*.

The Object tree is collapsed to show only the components of the model.

2. In the Object page, select *Air box*.
3. Click the right mouse button.
A pop-up menu appears.

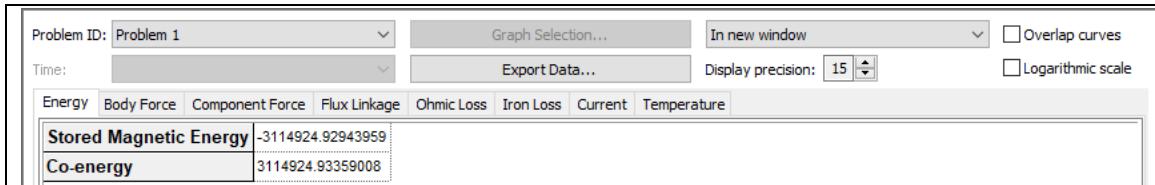
4. On the pop-up menu, toggle *Visible* so that the green check mark re-appears.



11 Solve

- On the Solve menu, click *Static 3D*.

The *Static 3D Solver Progress* dialog is displayed for a few minutes and, once the solution is complete, the Results window opens.



12 View the solution results

In this section, the force and torque on the rotating block is displayed in the Results window. A graph of the Z-component of the torque on the rotating block is created.

The shaded plot of **|B| smoothed** is viewed for *Problem 3*. An animation of the shaded plot over the eight solutions is created and viewed.

12.1 View the force on the rotating block

The Results window is automatically displayed when the solution is complete.

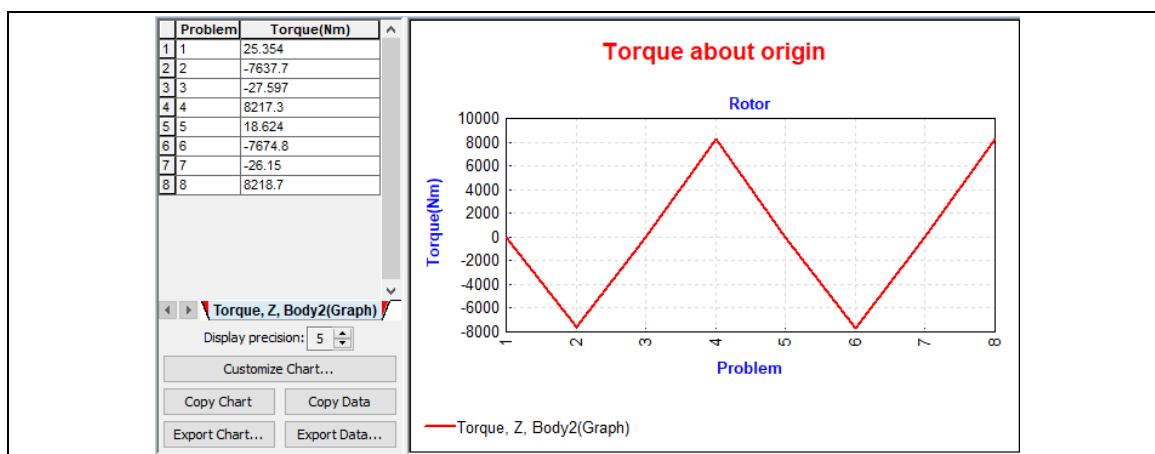
1. Select the *Body Force* tab on the Results window.
The *Force* page appears.
2. On the *Problem ID* drop down list, select *Problem 3*.
In Problem 3, the rotor is positioned at 90 degrees.
The Force values for Problem 3 are displayed. (The values may differ slightly depending on your computer.)

ID	Component(s)	Force				Torque			
		X	Y	Z	Magnitude	X	Y	Z	Magnitude
1	Core upper half + Core lower half + Permanent Magnet	116.340111822871	-52.5235717078875	54.6770666993697	138.86442606643	289.021817747734	27.5782452735184	81.9644332956766	301.682513699505
2	Rotor	-95.3251002830337	-3.16440832698613	-3.40008219804036	95.4381935232628	10.3692883445797	-34.0020700693691	-27.597011382669	45.0028659868511

12.2 Graph the Z-component of the torque

1. Click the mouse pointer in the *Z component of the torque vector* entry for the Rotor (highlighted in the image above).
2. Click (Graph Selection button).

A Graph view appears.



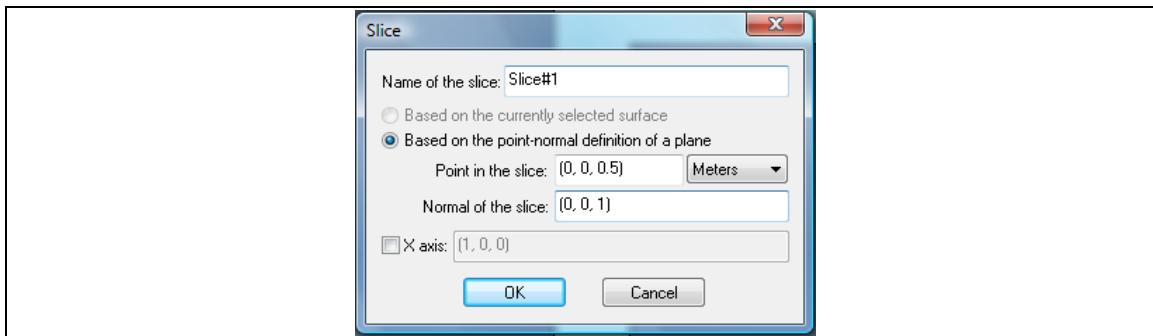
3. Switch back to the View window by clicking the "View 1" tab located at the bottom of the window.

13 View the shaded plot on a slice

13.1 Make a slice

1. On the Tools menu, click *New Slice*.

The *Slice* dialog box appears.



2. In the *Point in the slice* text box, enter **(0, 0, 0.5)**.
The point in the slice is described in global (3D) coordinates.
3. In the *Normal of the slice* text box, enter **(0, 0, 1)**.
This vector describes the plane perpendicular to the slice.
4. Click OK.
There is no need to set the X-axis in this case.

13.2 Hide the display of all the components

The shaded plot is visible on the surface of all the components toggled visible in the Object page. In this case, every component should be hidden from the display so that the slice is more easily viewed.

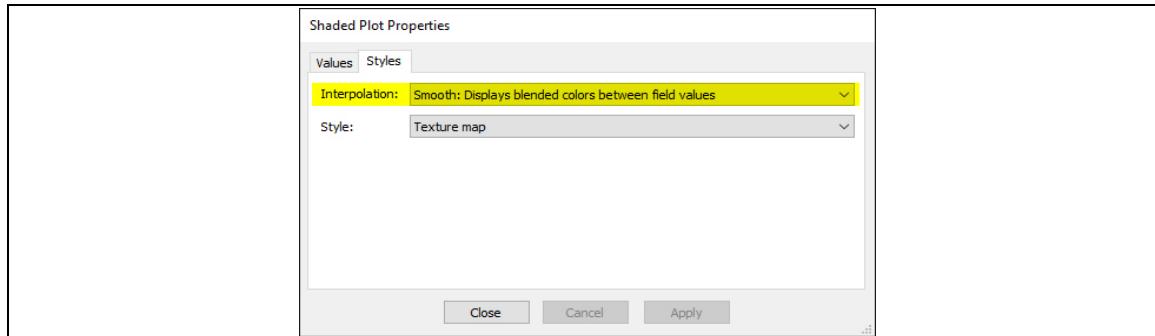
1. In the Object page, select Core Upper Half, Core Lower Half, Permanent magnet, Rotor, and Air box.
Tip Press the SHIFT key on your keyboard while selecting the components.
2. Click the right mouse button.
A pop-up menu appears.
3. On the pop-up menu, toggle *Visible* so that the check mark disappears.
On the Object page, a red X is displayed next to all the components.
All components are hidden from the display.

13.3 Set the color interpolation and style of the shaded plot

This procedure will set the default for shaded plots to smooth instead of discrete, which is the default.

1. On the View menu, click *Default Fields*.
2. On the Project Bar, select the *View* tab.
3. From the View tree, click *Shaded Plot*.
4. On the Edit menu, click *Properties*.

The Shaded Plot Properties page appears.



5. Click the Styles tab and then from the Interpolation drop-down menu, select "Smooth: Displays blended colors between field values".
6. Click OK.

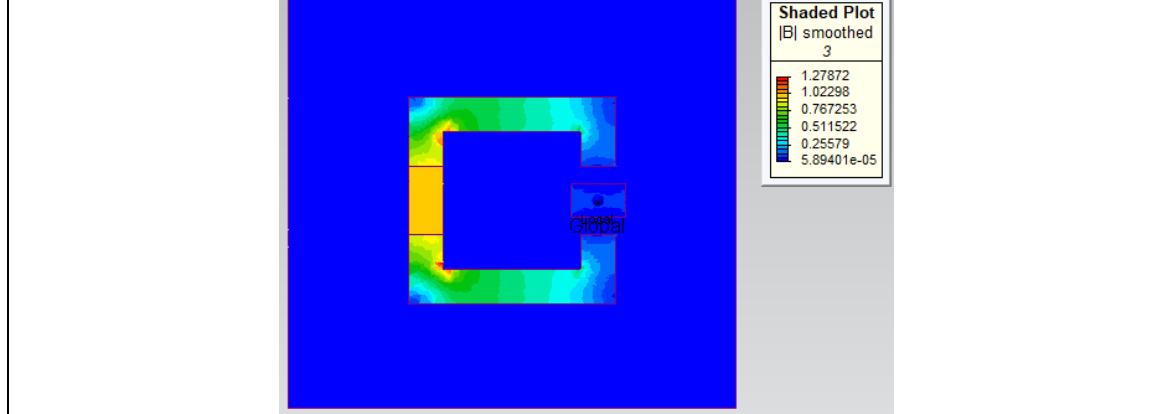
13.4 View the shaded plot

1. On the Project bar, select the *Field* tab.

The *Field* page opens.

2. From the *Problem ID* drop-down box, select **3**.
3. Select the *Shaded* tab.
4. In the *Fields to display* list, make sure that **|B| smoothed** is selected.
|B| smoothed is the default field.
5. Click the *Update View* button (near the bottom of the Field page).

The shaded plot on the slice is displayed. A color legend is displayed next to the shaded plot.



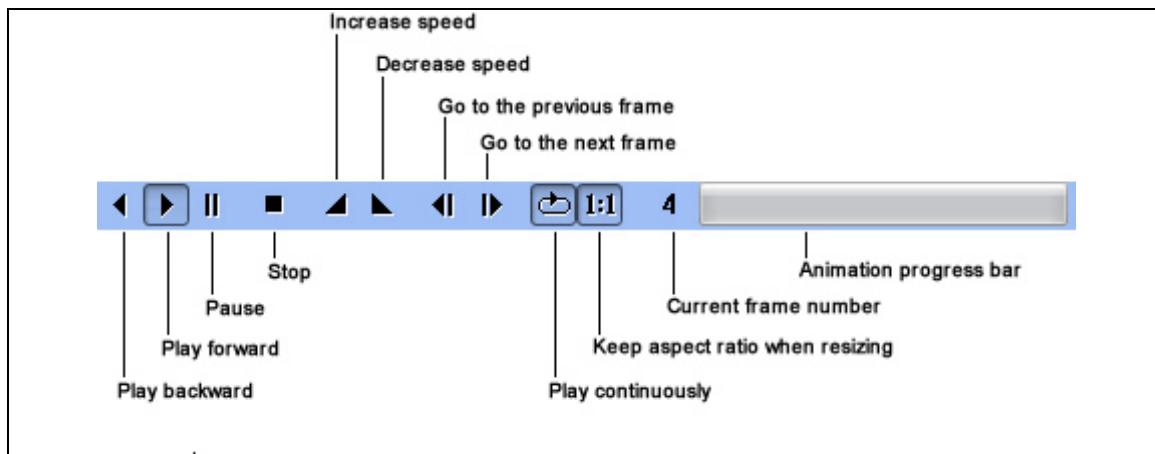
14 Animate the shaded plot

An animation is a series of “snapshots”, or frames, that are viewed as a moving image.

Note The color legend that is displayed during the animation represents the range of field values over the entire animation.

1. On the Field page, click  (Animate button).

Please wait a moment while the animation is created. When the animation is complete, an Animation window appears. The animation automatically plays in a loop. Use the Animation Control toolbar to control the playing of the animation.



2. Click the Stop button  when you are finished viewing the animation.

14.1 Save the animation

1. On the File menu, click *Save*.
The *Save As* dialog box appears.
2. In the *File Name* text box, enter **C-core with rotating block**.
The animation extension (.ban) is automatically added.
3. Click *Save*.
The animation is saved.
4. On the File menu, click *Close*.
The Animation window closes.

14.2 Save the model

You have now completed the tutorial.

1. On the File menu, click *Save*.
2. On the File menu, click *Close*.

15 Summary

In this tutorial, you parameterized the position of a component and viewed post-processing results across multiple solutions. The skills you learned include:

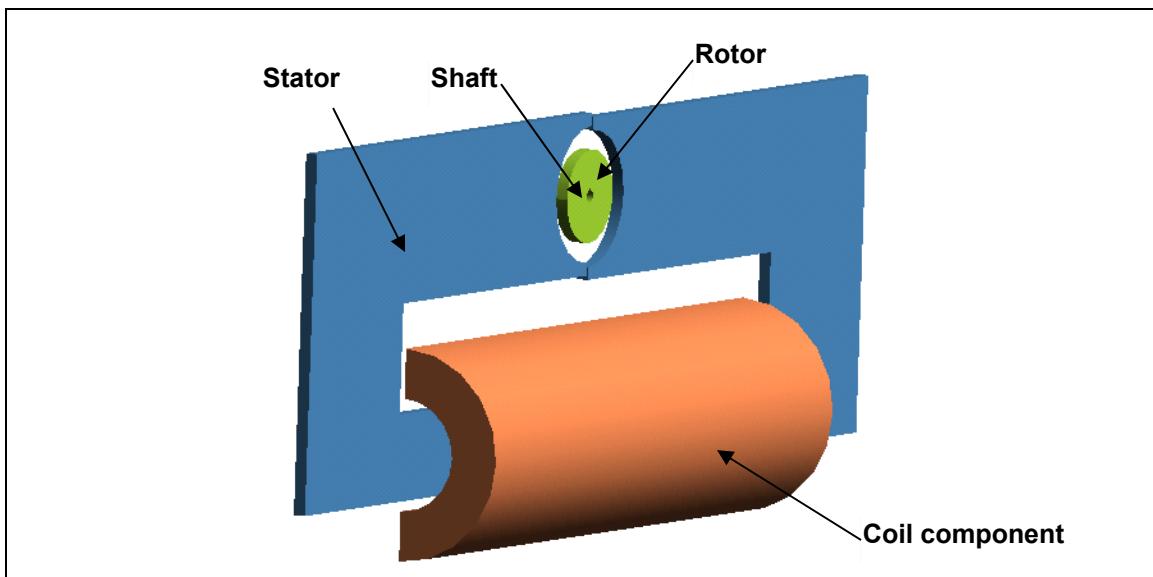
- Creating a user-defined parameter
- Rotating the geometric position of a component using parameterization
- Viewing instantiations of a model
- Viewing force and torque on a body
- Graphing the Z component of torque over multiple solutions
- Creating an animation of the shaded plot over multiple solutions

Tutorial #11

3D Transient with Motion
Permanent Magnet Stepper Motor

1 Modeling plan

The stepper motor used in this tutorial is made of a stator with eccentric pole faces and a rotor made of samarium cobalt permanent magnet that is magnetized in a fixed direction. The rotor rotates in steps of 180 degrees. Each step in the rotor is due to a short pulse from a current source. To obtain a unidirectional motion, the current pulse is alternating. In this tutorial, the response due to one pulse is examined. The damping due to bearing friction is taken into account in this simulation. Since there is a geometric and electromagnetic symmetry, only half of the device needs to be modeled. It should be noted that it is also possible to reduce the size of the problem further by modeling one side of the stator and rotor, and applying a periodic boundary condition. However, in this tutorial, we have not taken advantage of this symmetry condition.



2 Creating a new model

2.1 Opening a new model

- From your desktop, double-click the *Simcenter MAGNET icon*.

The Main window appears.

or

- If Simcenter MAGNET is already running, on the File menu, click *New* to open a new model.

2.2 Name the model

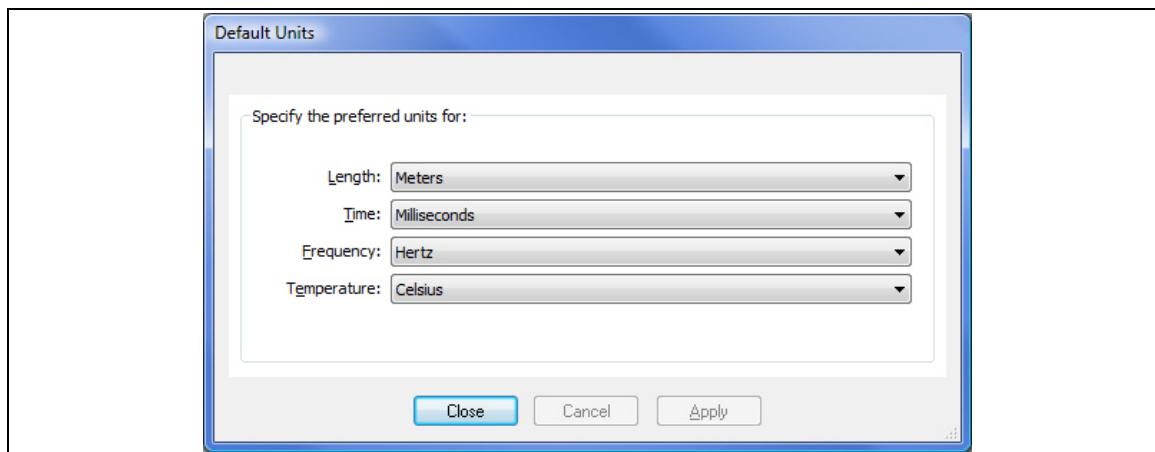
By default, Simcenter MAGNET assigns a name to the model (e.g., Simcenter MAGNET1) every time a new model is opened. As long as the application remains open, each new model number increments by one (e.g., if the new model you have opened is the fourth one in this session, Simcenter MAGNET would assign the name Simcenter MAGNET4). You can choose to retain this name, although it is recommended that you give the model a distinct name.

1. On the File menu, click *Save As*.
2. In the Save As dialog box, enter **PM Stepper Motor** as the name of the model.
3. Choose the drive and directory where you want to place the model.
4. Click *Save As*.

3 Setting up the working environment

3.1 Initial Settings

Each new model reverts to the Simcenter MAGNET default settings for the preferred units for length, time, frequency, and temperature. For our model, we are going to change only the preferred units for length to *Millimeters* and accept all the other defaults.



1. On the Tools menu, click *Set Units*.
The Default Units dialog appears.
2. From the *Length* drop down list, select *Millimeters*.
3. Click OK.

3.2 Set view to update automatically

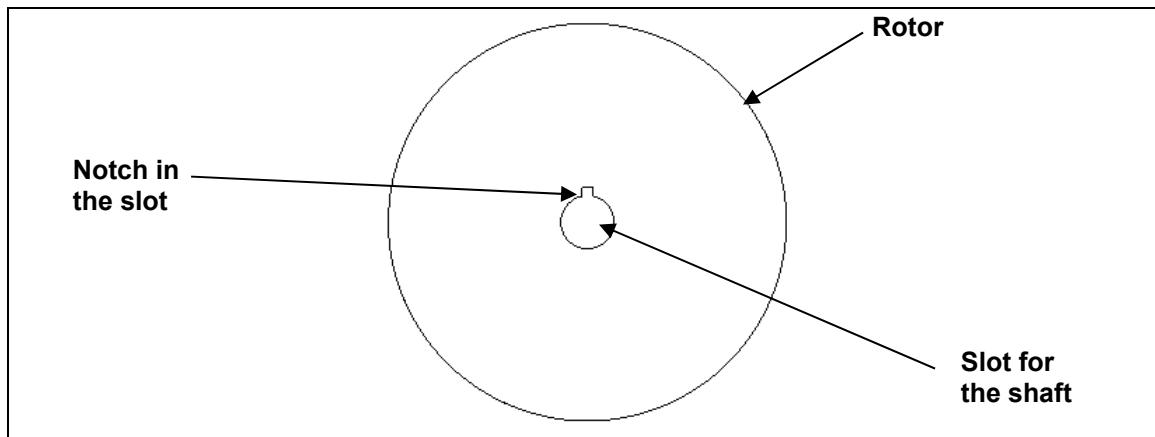
Use of this feature resets the view automatically to include all of the model's geometry, as it is being drawn.

- On the View menu, click *Update Automatically*.

4 Build the geometric model and set up the problem for the Rotor, Stator, Air box, and the Coil

4.1 Draw the geometric model of the rotor

Incorporated into this geometric model of the rotor is the slot for the shaft of the Stepper Motor.



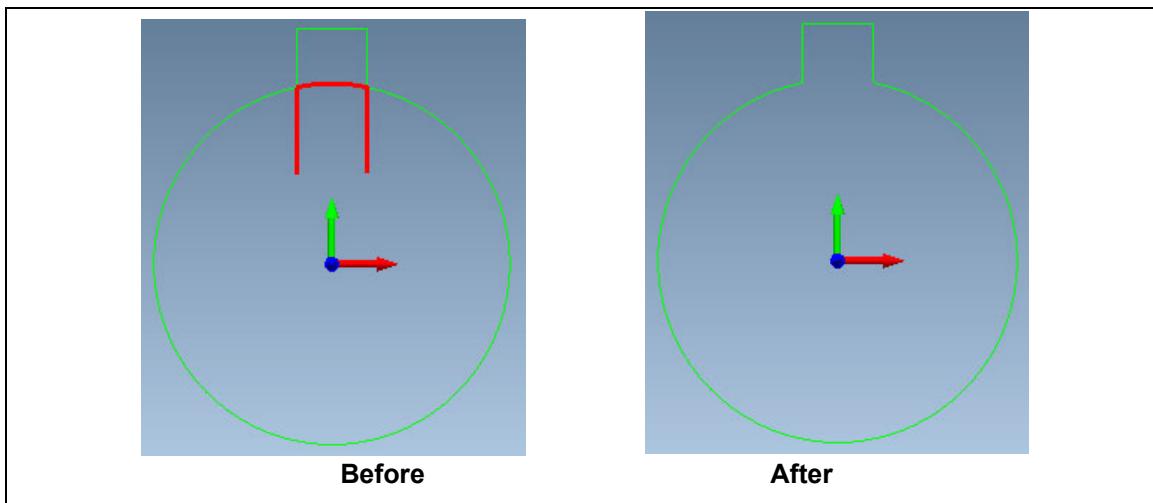
1. On the Tools menu, click *Keyboard Input Bar*.
2. On the Draw menu, click *Circle (Center, Radius)*.
3. In the Keyboard Input bar, enter the following coordinates to draw the Rotor:

Center point	0, 0	Press ENTER
Point on the radius of the circle	0.75, 0	Press ENTER
4. In the Keyboard Input bar, enter the following coordinates to draw the slot for the shaft:

Center point	0, 0	Press ENTER
Point on the radius of the circle	0.1, 0	Press ENTER
5. On the Draw menu, click *Line*.
6. In the Keyboard Input bar, enter the following coordinates to draw the notch in the shaft:

Start coordinates	-0.02, 0.05	Press ENTER
mid coordinates	-0.02, 0.13	Press ENTER
mid coordinates	0.02, 0.13	Press ENTER
End coordinates	0.02, 0.05	Press ENTER, and then ESC
7. On the View toolbar, click (Examine Model).
- 8.
9. Hold down the CTRL key and the left mouse button to form a rectangular box around the Rotor slot's circle and the notch's lines.
10. On the Edit menu, click *Select Construction Slice Edges*.
11. Keeping the mouse button clicked, drag the cursor to form a rectangle around the Rotor slot's circle and the notch's lines.
All of the Construction Slice edges are selected.
12. On the Draw menu, click *Segment Edges*.
All of the edges that share coordinates are segmented.

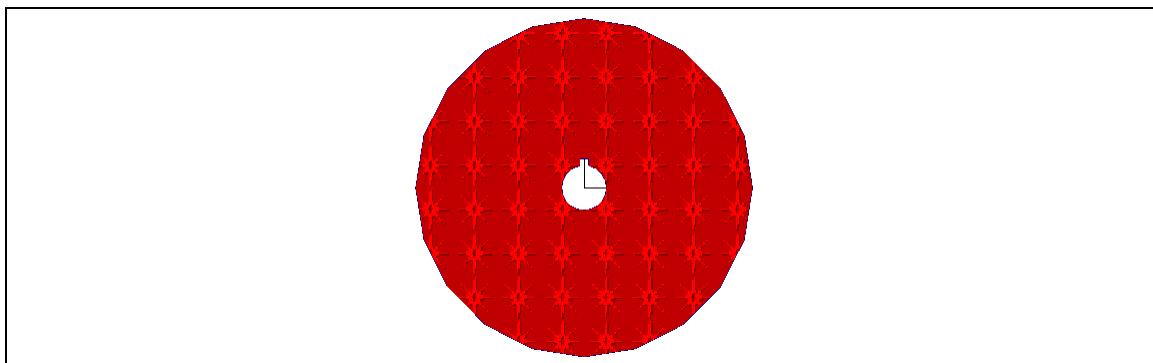
13. Keeping the CTRL button down, click the interior edges as shown in the *Before* illustration.



14. On the Edit menu, click *Delete*.

4.2 Make the Rotor component

1. On the View menu, click *Update Automatically*.
2. On the Edit menu, click Select Construction Slice Surfaces.
3. Select the interior region of the geometric model you have drawn, as shown below.



4. On the Model menu, click *Make Component in a Line*, and enter the following values:
 - Name: **Rotor**
 - Material: **Samarium Cobalt**
 - Material Direction:
 - Type: **Uniform**
 - Direction: **(0,0,1)**
 - Distance **0.2 Millimeters**
5. Click OK.

4.3 Delete Construction Slice edges

Since the edges on the construction slice remain even after being swept into a component, we must remove them before proceeding to the next step of the tutorial.

1. On the Edit menu, click Select Construction Slice Edges.
2. On the Edit menu, click *Select All*.
3. Press the Delete key.

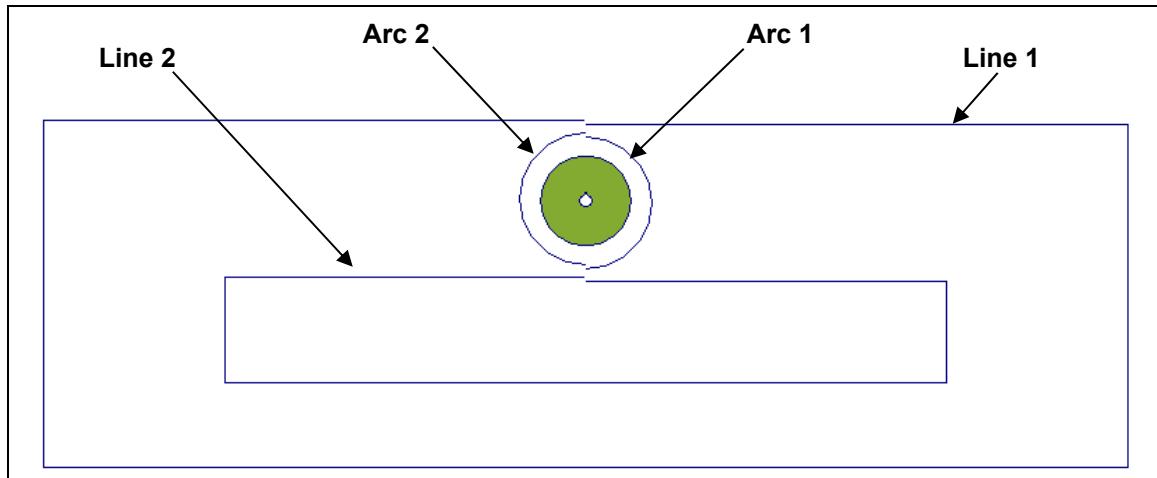
4.4 Setting the Rotor's magnetization direction in the initial position

1. From the Object page, select the Rotor component.
2. On the Edit menu, click *Properties*.
3. Select the Parameters tab.
4. Scroll down to MaterialDirection parameter.
5. In the Expression column, type `[cos(67%deg), sin(67%deg), 0]` and then press TAB.

Parameter	Type	Expression (default units are SI)
MaterialConductivityScaleFactor	Number	
MaterialDirection	Array	<code>[cos(67%deg), sin(67%deg), 0]</code>

6. Click OK.

4.5 Draw the geometric model of the Stator



1. On the Draw menu, click *Arc (Center, Start, End)*.

2. In the Keyboard Input bar, enter the following coordinates to draw Arcs 1 and 2:

Arc 1

Center coordinates	0, -0.035	Press ENTER
Start coordinates	0, -1.135	Press ENTER
End coordinates	0, 1.065	Press ENTER

Arc 2

Center coordinates	0, 0.035	Press ENTER
Start coordinates	0, 1.135	Press ENTER
End coordinates	0, -1.065	Press ENTER

3. On the Draw menu, click *Line*.

4. In the Keyboard Input bar, enter the following coordinates to draw the Lines 1 and 2:

Line 1

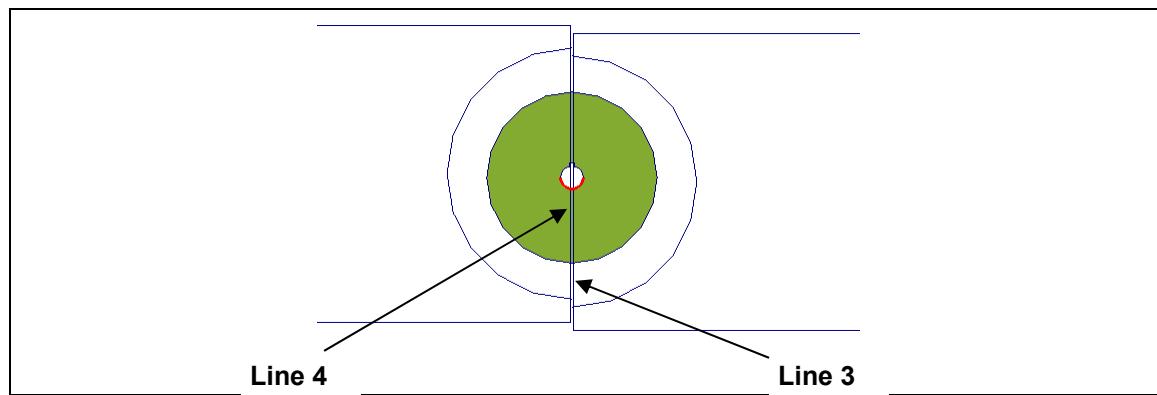
Start coordinates	0.01, 1.265	Press ENTER
mid coordinates	9, 1.265	Press ENTER
mid coordinates	9, -4.435	Press ENTER
mid coordinates	-9, -4.435	Press ENTER
mid coordinates	-9, 1.335	Press ENTER
End coordinates	-0.01, 1.335	Press ENTER, and then ESC

Line 2

Start coordinates	0.01, -1.335	Press ENTER
mid coordinates	6, -1.335	Press ENTER
mid coordinates	6, -3.035	Press ENTER
mid coordinates	-6, -3.035	Press ENTER
mid coordinates	-6, -1.265	Press ENTER
End coordinates	-0.01, -1.265	Press ENTER, and then ESC

5. On the View toolbar, click  (Examine Model).
 6. Hold down the CTRL key and the left mouse button to form a rectangular box around the arcs surrounding the Rotor component.
 7. On the Draw menu, click *Line*.

8. The next step that is needed to complete the geometric modeling of the Stator -- drawing two lines (Line 3 and Line 4) that serve to create a closed surface -- is accomplished by entering, in the Keyboard Input bar, the following coordinates:



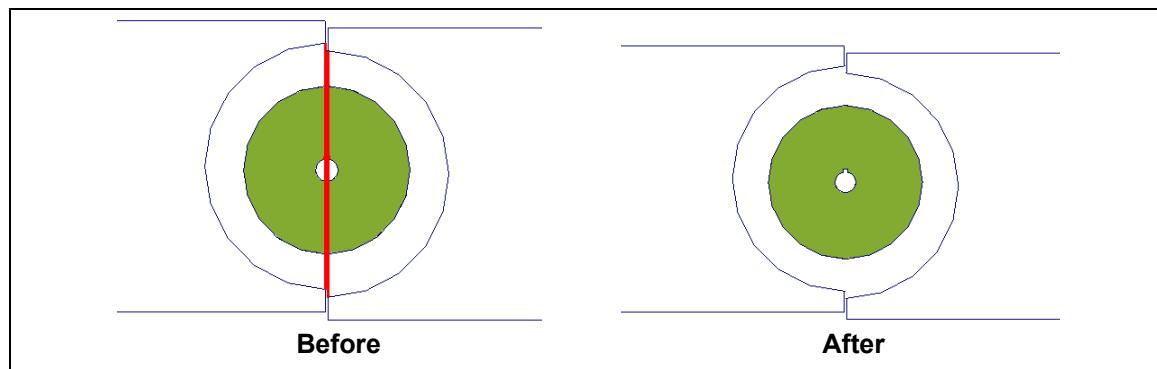
Line 3

Start coordinates	0.01, -1.335	Press ENTER
End coordinates	0.01, 1.265	Press ENTER, and then ESC

Line 4

Start coordinates	-0.01, -1.265	Press ENTER
End coordinates	-0.01, 1.335	Press ENTER, and then ESC

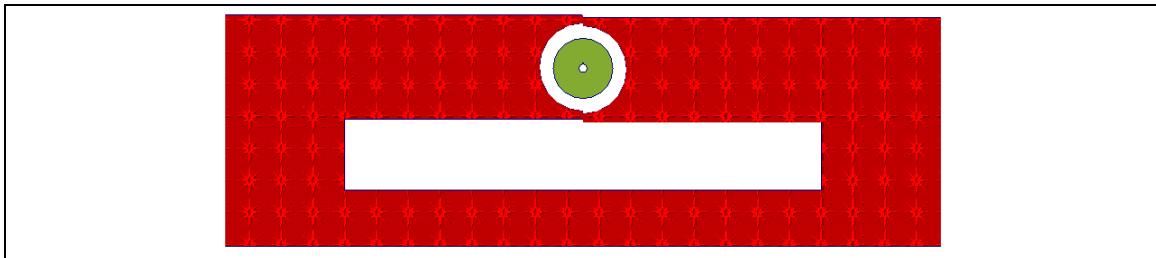
9. On the Edit menu, click Select Construction Slice Edges.
10. Keeping the mouse button clicked, drag the cursor to form a rectangle around the arcs surrounding the Rotor component.
All of the Construction Slice edges are selected.
11. On the Draw menu, click *Segment Edges*.
All of the edges that share coordinates are segmented.
12. Keeping the CTRL button down, click the interior edges as shown in the *Before* illustration.



13. On the Edit menu, click *Delete*.
14. On the File menu, click *Save*.

4.6 Make the Stator component

1. On the View menu, click *Update Automatically*.
2. On the Edit menu, click Select Construction Slice Surfaces.
3. Select the interior region of the geometric model you have drawn, as shown below.



4. On the Model menu, click *Make Component in a Line*, and enter the following values:
 - Name: **Stator**
 - Material: **CR10: Cold rolled 1010 steel**
 - Distance **0.2 Millimeters**
5. Click OK.

4.7 Delete Construction Slice edges

Since the edges on the construction slice remain even after being swept into a component, we must remove them before proceeding to the next step of the tutorial.

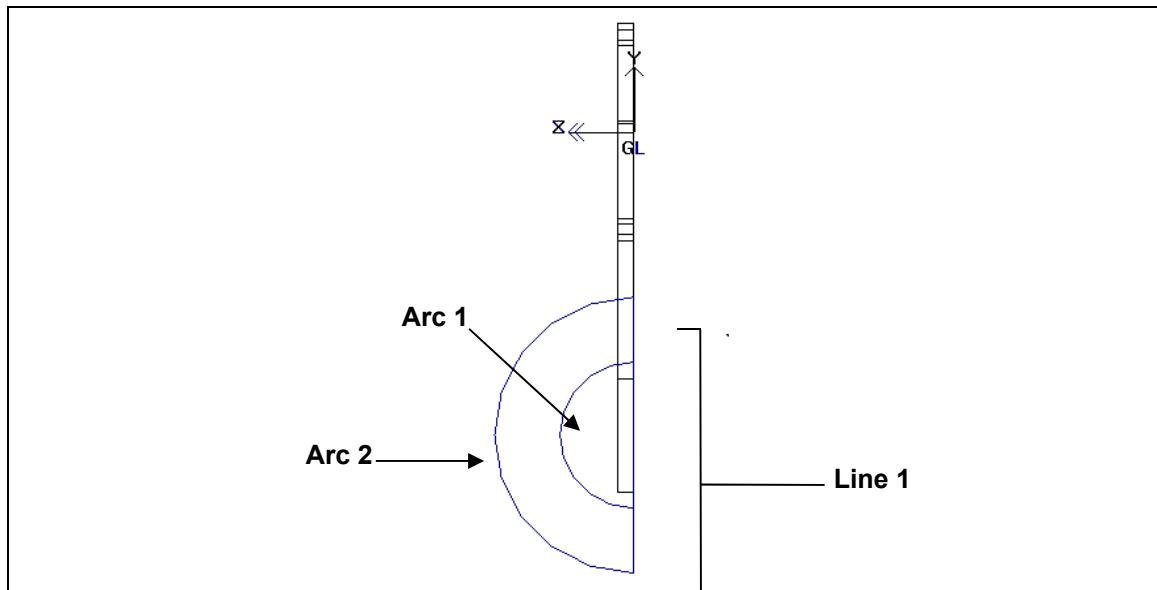
1. On the Edit menu, click *Select Construction Slice Edges*.
2. On the Edit menu, click *Select All*.
3. Press the Delete key.

4.8 Move the construction slice

Before drawing the geometry of the next component, the construction slice must be moved.

1. On the Object page, select Stator, Face#10.
2. On the Draw menu, click *Move Construction Slice*.
3. In the Move Construction Slice dialog box, make sure that *To The Currently Selected Surface* is selected.
4. Click OK.
5. On the Draw menu, click *Move Construction Slice*.
6. In the Move Construction Slice dialog box, select *Along a line* and then type **0.5** in the *Distance* text box.
7. Click OK.

4.9 Draw the geometric model of the Coil component



1. On the View menu, click *Wireframe Model*.
2. On the View menu, click *Preset Views*, and then click *Positive X Axis*.
3. On the Draw menu, click *Arc (Center, Start, End)*.
4. In the Keyboard Input bar, enter the following coordinates to draw Arcs 1 and 2:

Arc 1

Center coordinates	0, -3.735	Press ENTER
Start coordinates	0, -4.635	Press ENTER
End coordinates	0, -2.835	Press ENTER

Arc 2

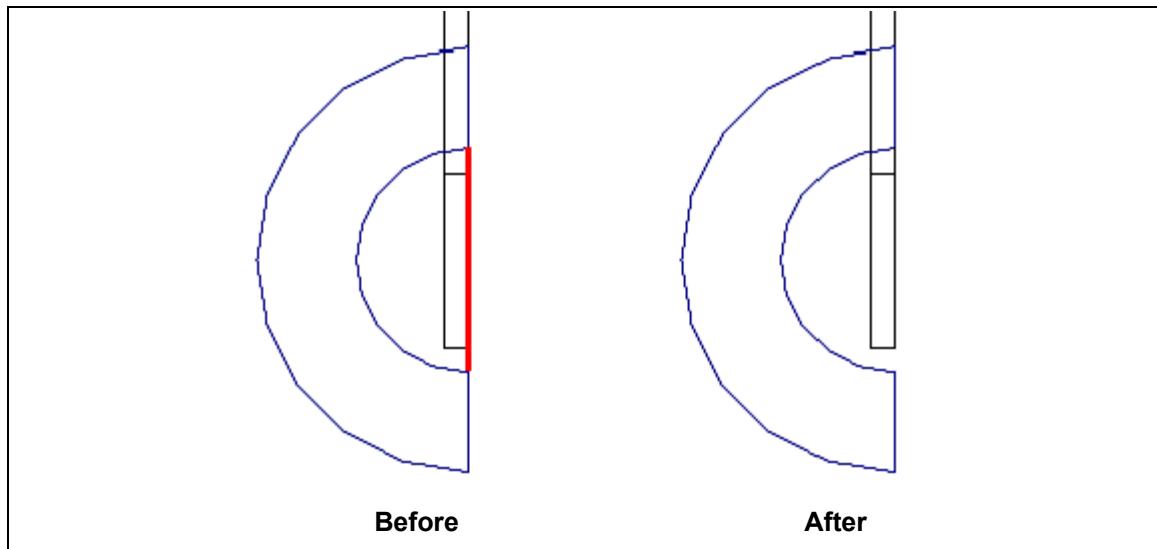
Center coordinates	0, -3.735	Press ENTER
Start coordinates	0, -5.435	Press ENTER
End coordinates	0, -2.035	Press ENTER

Note To complete the geometric modeling of the Coil component-- drawing a line (Line 1) that serves to create a closed surface for the two arcs that were just drawn -- is accomplished by the steps that follow:

5. On the Draw menu, click *Line*.
6. On the Draw menu, click *Snap Modes*, and then click *Endpoint*.
7. Using the mouse pointer, click once at each endpoint of Arc 2.
8. On the Edit menu, click *Select Construction Slice Edges*.
9. Keeping the mouse button clicked, drag the cursor to form a rectangle around the two arcs and the line.

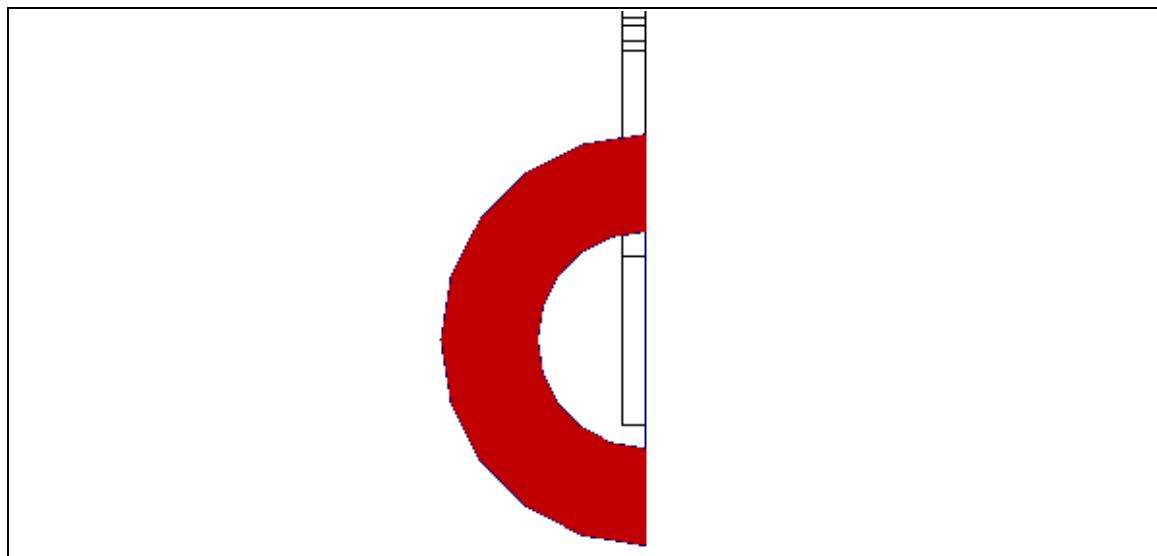
All of the Construction Slice edges are selected.

10. On the Draw menu, click *Segment Edges*.
All of the edges that share coordinates are segmented.
11. Click the interior edge as shown in the “Before” illustration.
12. On the Edit menu, click *Delete*.



4.10 Make the Coil component

1. On the Edit menu, click *Select Construction Slice Surfaces*.
2. Select the interior region of the geometric model you have drawn, as shown below.



3. On the Model menu, click *Make Component in a Line*, and enter the following values:
 - Name: **Coil component**
 - Material: **Copper: 5.77e7 Siemens/meter**
 - Distance **11.0 Millimeters**
4. Click OK.

4.11 Delete Construction Slice edges

Since the edges on the construction slice remain even after being swept into a component, we must remove them before proceeding to the next step of the tutorial.

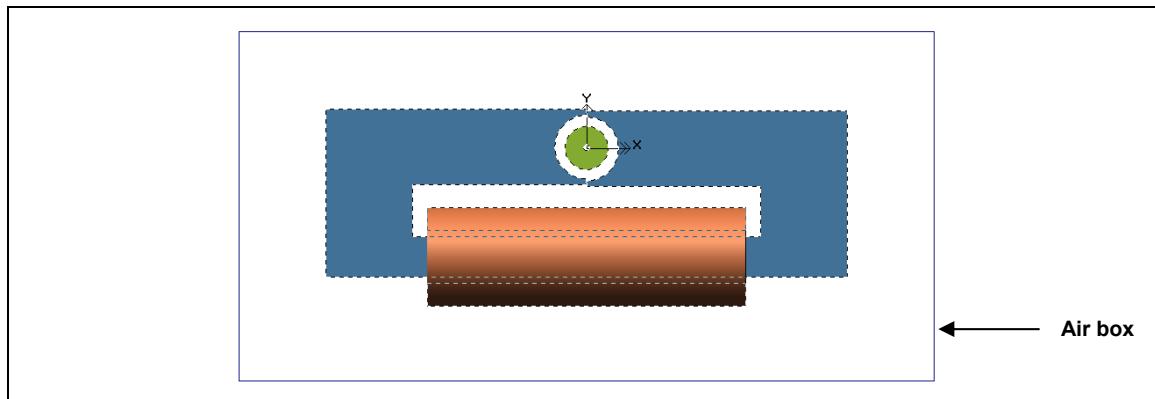
1. On the Edit menu, click Select Construction Slice Edges.
2. On the Edit menu, click *Select All*.
3. Press the Delete key.

4.12 Resetting the construction slice position

Before drawing the geometry of the next component, the construction slice must be reset to its original position.

1. On the Object page, select Rotor, Face#1 (Start Face).
2. On the Draw menu, click *Move Construction Slice*.
3. In the Move Construction Slice dialog box, make sure that To The Currently Selected Surface is selected.
4. Click OK.

4.13 Draw the geometric model of the Air box component



1. On the View menu, click *Update Automatically*.
2. On the View menu, click *Solid Model*.
3. On the View menu, click *Preset Views*, and then click *Positive Z Axis*.
4. On the Draw menu, click *Line*.
5. In the Keyboard Input bar, enter the following coordinates:

Start coordinates	-12, 4	Press ENTER
mid coordinates	12, 4	Press ENTER
mid coordinates	12, -8	Press ENTER
mid coordinates	-12, -8	Press ENTER
End coordinates	-12, 4	Press ENTER, and then ESC

4.14 Make the Air box component

1. On the Edit menu, click Select Construction Slice Surfaces.
2. Select the interior region of the geometric model you have drawn.
3. On the Model menu, click *Make Component in a Line*, and enter the following values:
 - Name: **Air box**
 - Material: **AIR**
 - Distance **4.0 Millimeters**
4. Click OK.

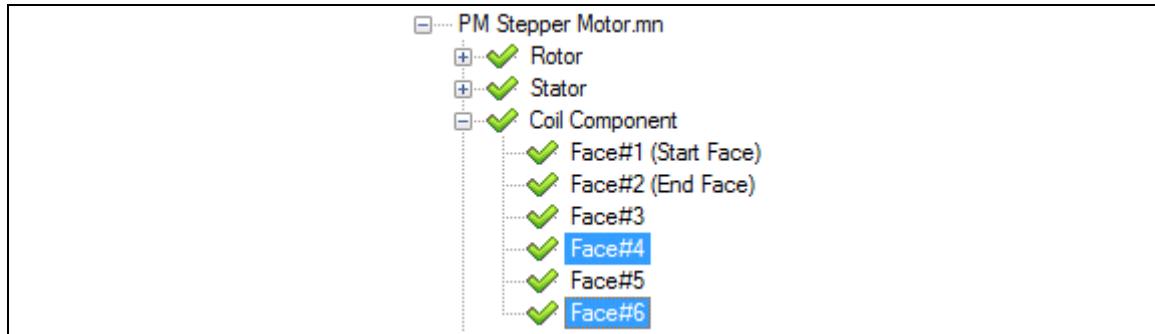
4.15 Delete Construction Slice edges

Since the edges on the construction slice remain even after being swept into a component, we must remove them before proceeding to the next step of the tutorial.

1. On the Edit menu, click Select Construction Slice Edges.
2. On the Edit menu, click *Select All*.
3. Press the Delete key.

5 Make the coil

1. From the Object page, click on the (+) sign of Coil component.
The faces of the component are displayed.
2. Keeping the CTRL key down, select Face#4 and Face#6 of the Coil component.



3. On the Model menu, click *Make Multi-Terminal Coil*.
Coil#1 is created and listed in the Object page.
4. From the Object page, select Coil#1.
5. On the Edit menu, click *Properties*.
The Coil Properties dialog appears.
6. In the Coil Attributes page, do the following:
 - for *Type*, select **Stranded**
 - for *No. of Turns*, type **10000**
 - for *Source*, select **CurrentDriven**

7. Select the Waveform tab and do the following:

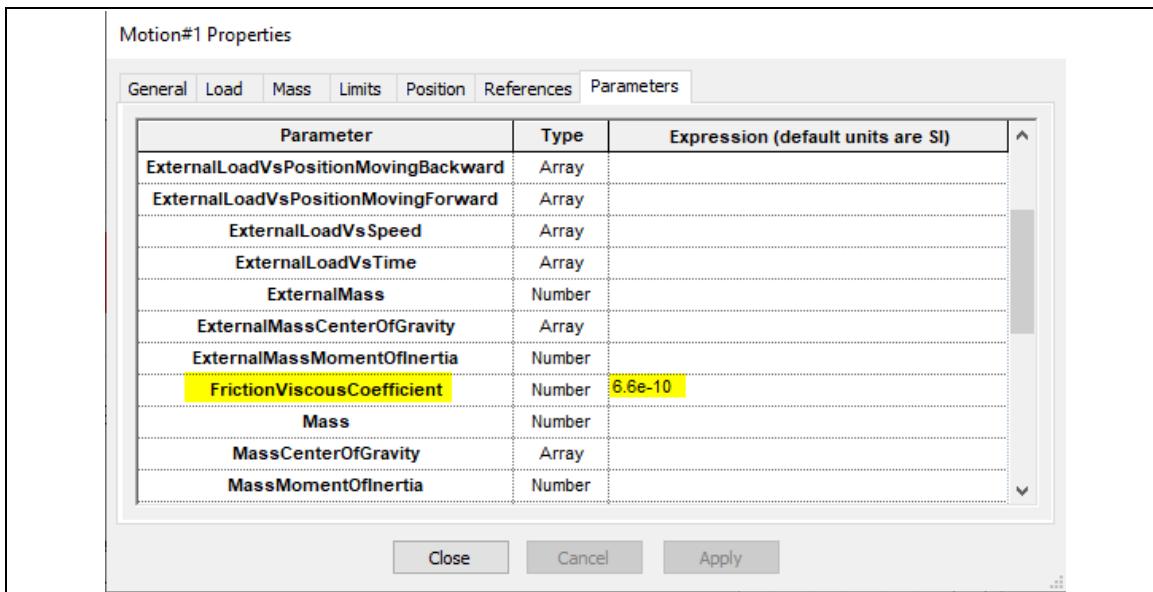
- for Type, select Piecewise linear
- enter the following values:

	<u>Time (s)</u>	<u>Value</u>
1	0	0
2	1e-006	0.0006
3	0.004	0.0006
4	0.004001	0
5	0.01	0

8. Click OK.

6 Make the motion component

For the purpose of this analysis, we will need to set the motion component to be load driven. This means that the velocity of the motion component is not known and that it varies with time as a result of the generated electromagnetic field.



1. From the Object page, select *Rotor*.

2. On the Model menu, click *Make Motion Component*.

Motion#1 is created and listed in the Object page, and the Motion Component dialog appears.

3. Select the *Parameters* tab.

4. Scroll down to FrictionViscousCoefficient parameter (*highlighted in the image above*).

5. In the Expression column, type **6.6e-10** and then press TAB.

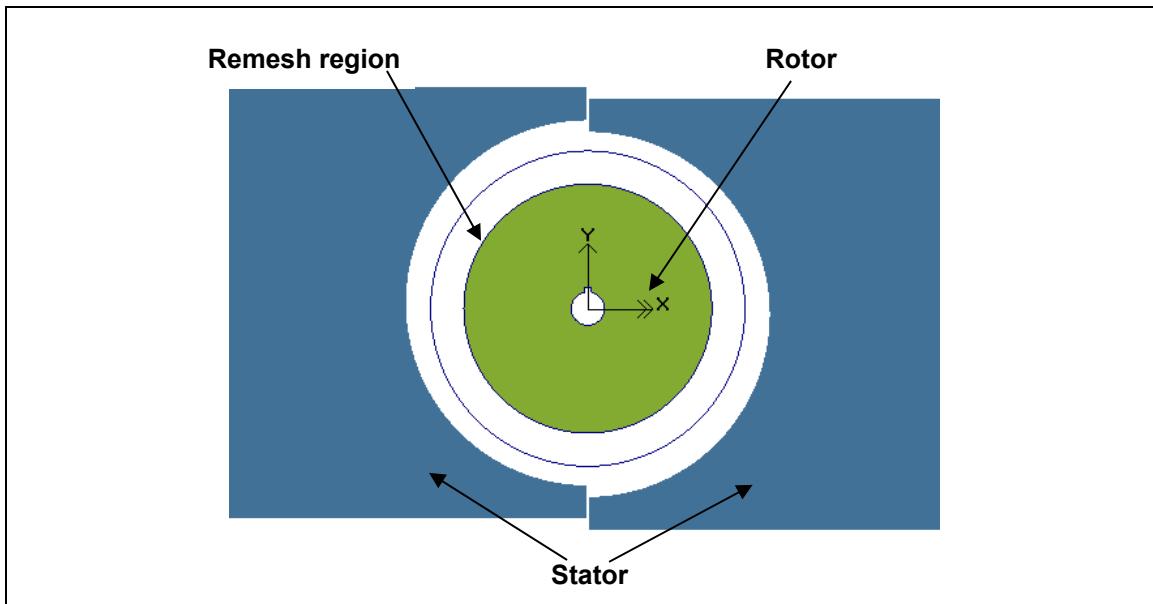
This load represents the bearing friction.

6. Click OK to accept all other defaults.

The motion component (Source type: Load Driven / Motion type: Rotary) is created.

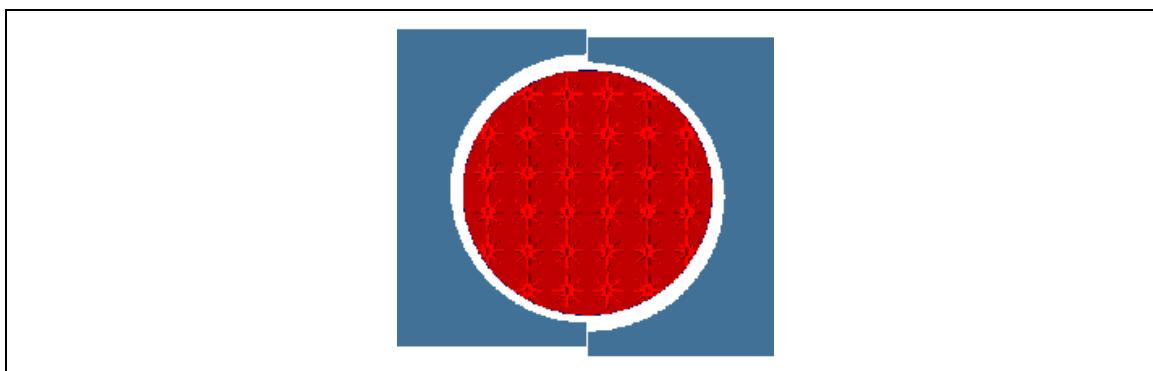
6.1 Create a Remesh Region component

Creating this component re-meshes the region surrounding the moving component. Re-meshing this relatively small and simple region is quick, and with this approach, no additional constraint equations need to be solved, which keeps solution times short and memory requirements low.



1. On the Draw menu, click *Circle (Center, Radius)*.
2. In the Keyboard Input bar, enter the following coordinates to draw the Remesh Region:

Center point	0, 0	Press ENTER
Point on the radius of the circle	1, 0	Press ENTER
3. On the Edit menu, click Select Construction Slice Surfaces.
4. Select the interior region of the geometric model you have drawn, as shown below.



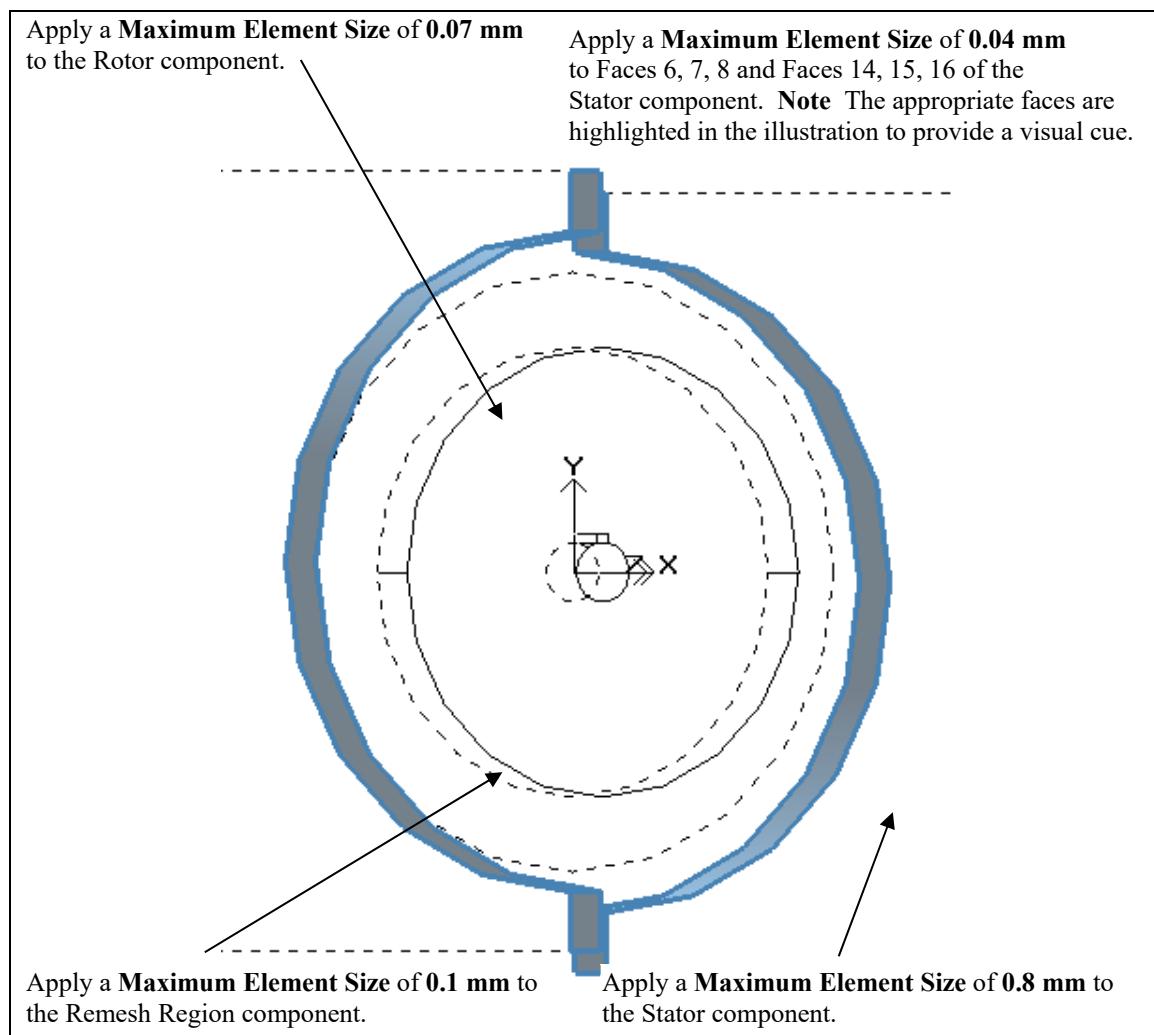
5. On the Model menu, click *Make Component in a Line*, and enter the following values:
 - Name: **Remesh Region**
 - Material: **AIR**
 - Distance: **0.3 Millimeters**
6. Click OK.

6.2 Modify the mesh

In the 3D finite element method of analysis, the model is divided into a mesh of tetrahedral-shaped elements. Each element is defined by four vertices (nodes). The vector field inside each element is represented by a polynomial with unknown coefficients. The finite element analysis is the solution of the set of equations for the unknown coefficients.

The accuracy of the solution depends upon the nature of the field and the size of the mesh elements. In regions where the direction or magnitude of the field is changing rapidly, high accuracy requires small elements. One method of increasing mesh density is to set the maximum element size for a component volume or specific faces of a component. The following procedure will demonstrate this method.

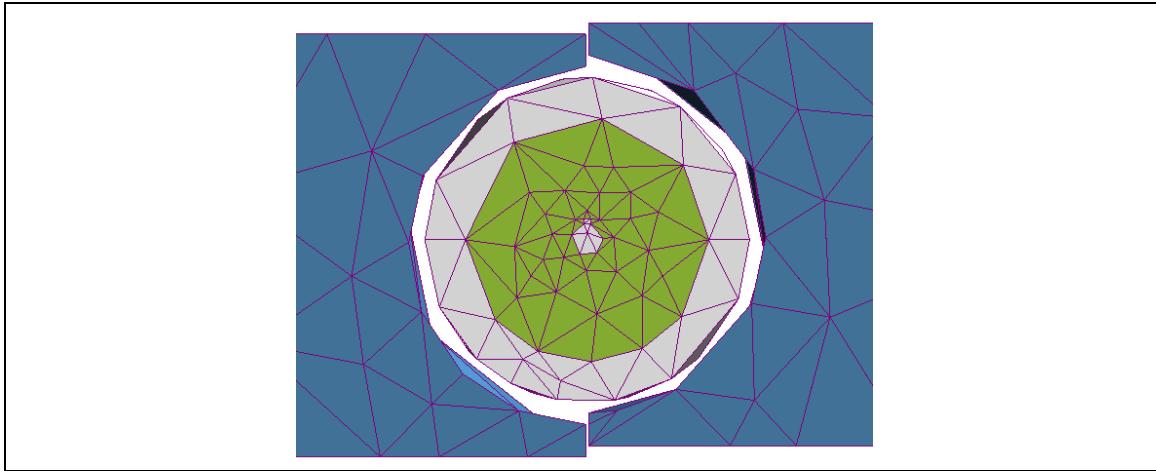
The maximum element size modifications for this model are shown in the diagram below.



Note Before changing the maximum element size, the default 3D initial mesh can be viewed. For the purpose of this exercise only, we will hide the Air box component to allow an unobstructed view of the mesh.

1. Select Air box from the Object page.
2. Right-click and then click *Visible*.
The component is hidden from view. A red “X” appears by the Air box component’s name in the Object page.
3. On the View menu, click *Preset Views*, and then *Negative Z Axis*.
4. On the View toolbar, click  (Examine Model).
5. Hold down the CTRL key and the left mouse button to form a rectangular box around the Rotor and Remesh Region, including the part of the Stator that envelops these two components.
6. On the View menu, click *Initial 3D Mesh*.

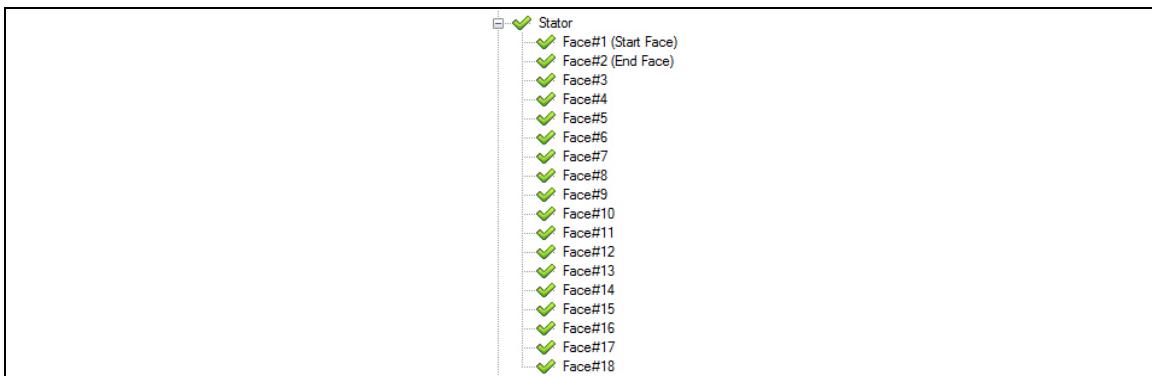
The initial 3D mesh appears in the View window. The mesh should look like the illustration shown below.



7. On the View menu, click *Solid Model*.
8. Select the **Rotor** component in the Object page.
9. On the Edit menu, click *Properties*, and then select the Mesh tab.
10. Click inside the Maximum element size checkbox, and then type **0.07** in the text box.
11. Click *Apply*.
12. Select the **Stator** component in the Object page.
13. Click inside the Maximum element size checkbox, and then type **0.8** in the text box.
14. Click *Apply*.
15. Select the **Remesh Region** component in the Object page.
16. Click inside the Maximum element size checkbox, and then type **0.1** in the text box.
17. Click *OK*.

18. From the Object page, click on the (+) sign of Stator.

The faces of the Stator component are displayed.



19. Select Face#6 of the Stator component.

20. On the Edit menu, click *Properties*.

The Face Properties dialog appears.

21. Select the Mesh tab.

22. Make sure that the Maximum element size checkbox is checked, and then type **0.04** in the text box.

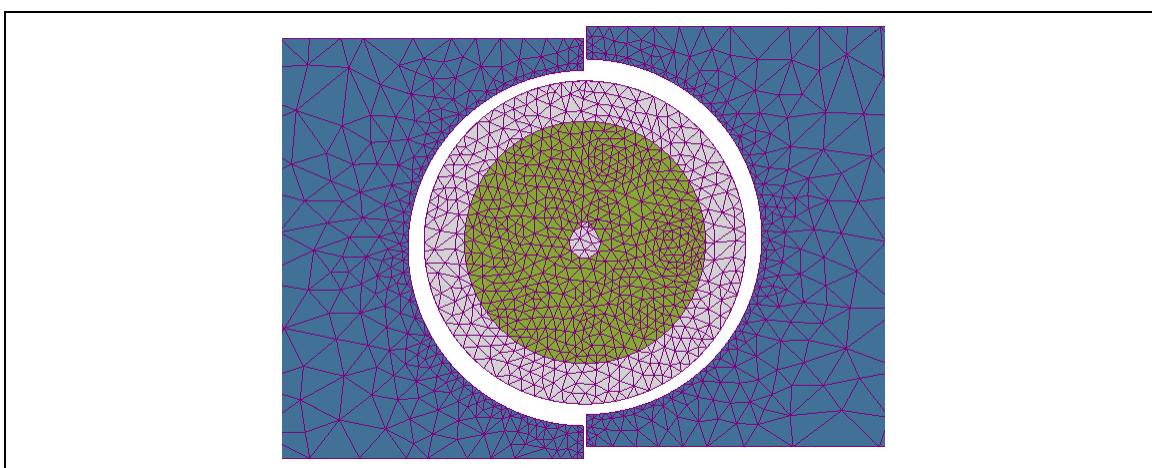
23. Click *Apply*.

24. Keeping the Face Properties dialog open, select Face#7 of the Stator component and repeat steps 22 and 23. Do the same for Face#8, Face#14, Face#15, and Face#16.

25. Click OK to close the Face Properties dialog.

26. On the View menu, click *Initial 3D Mesh*.

The initial 3D mesh (with the modifications) appears in the View window and should look like the illustration shown below.



27. On the View menu, click *Solid model*.

28. On the Object page, toggle the visibility of the Air box component so that it is visible. (Right-click the component and toggle Visible on the pop-up menu.).

The check symbol next to the Air box component indicates that it is visible.

7 Generating the Transient 3D with Motion field solution

7.1 Set the Transient options

1. On the Solve menu, click *Set Transient Options*.

The Set Transient Options dialog appears.

2. Make sure that Fixed Interval is selected as the Time Step Method, and then make the following modifications for Time:

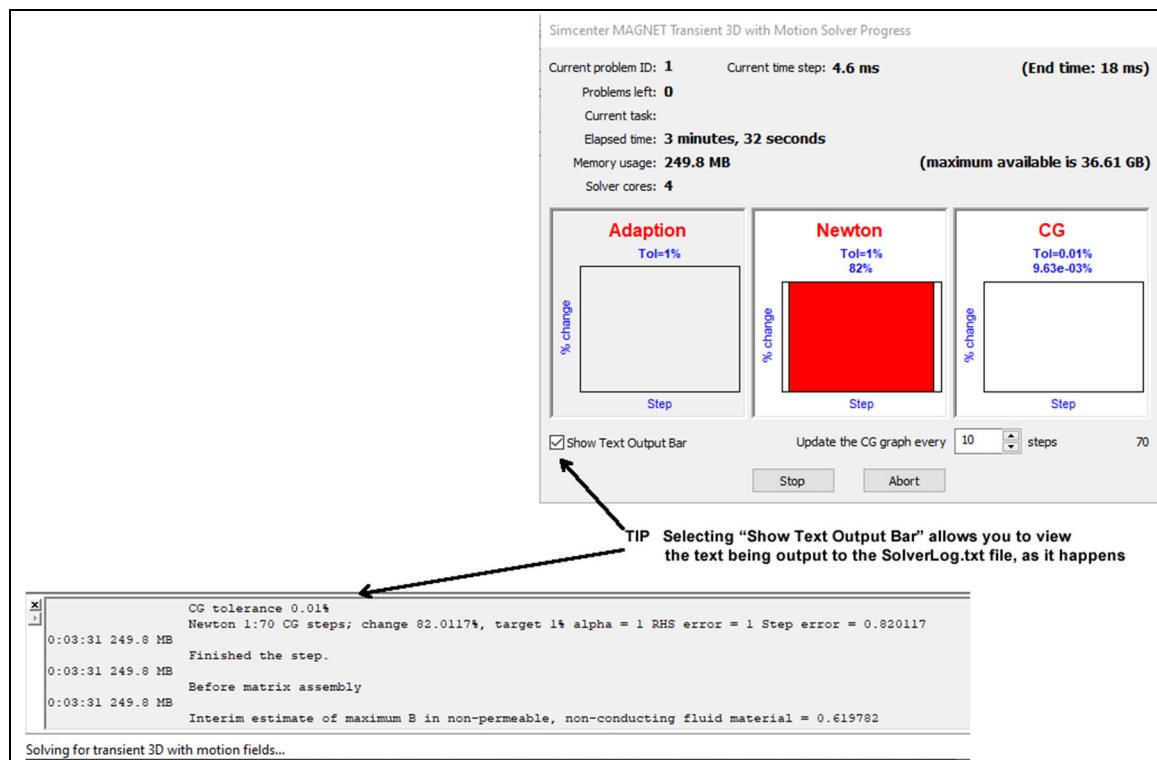
- Start = **0 Milliseconds**
- Stop = **18 Milliseconds**
- Step = **0.2 Milliseconds**

3. Click OK.

7.2 Solve the model

- On the Solve menu, click *Transient 3D with Motion*.

The *Transient 3D with Motion Solver* Progress dialog appears. **The solving process for this particular problem may take some time to complete.**



8 Analyzing the results

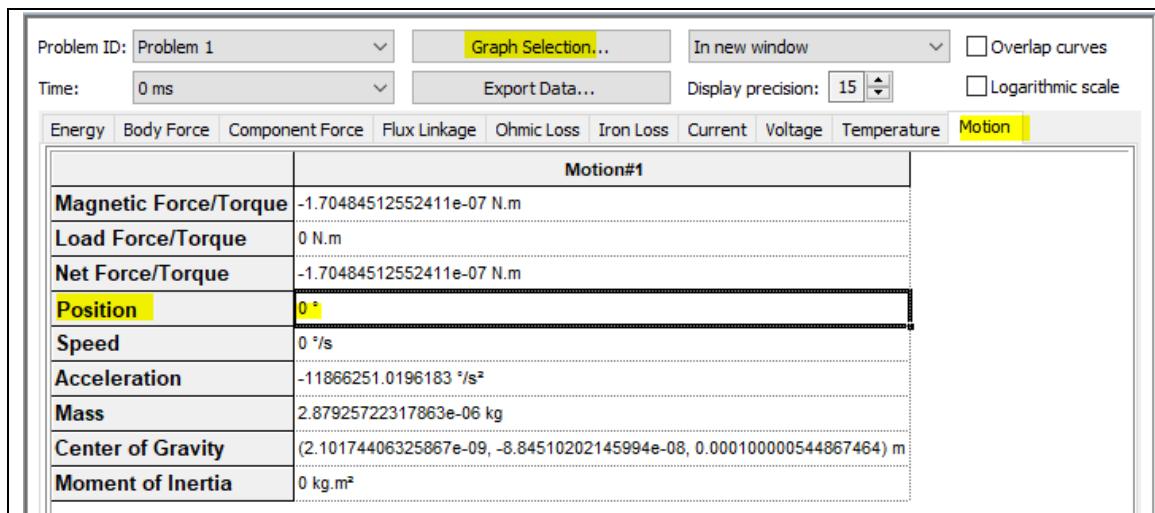
In this section, we will examine results for the motion component by selecting the entries for position, magnetic torque, velocity, and acceleration, and graphing the results for each one. In addition, we will create an animation of the shaded plot of $|B|$ with a view that is focused on the motion component.

Once the solution process is complete, it is possible to examine the speed and magnetic torque of the rotor over the entire time, from standstill to synchronous speed.

8.1 Graph the position, magnetic torque, velocity, and acceleration of the motion component

The Results window is automatically displayed when the solution is complete. The results in the illustration below are those from the first time step (0 ms).

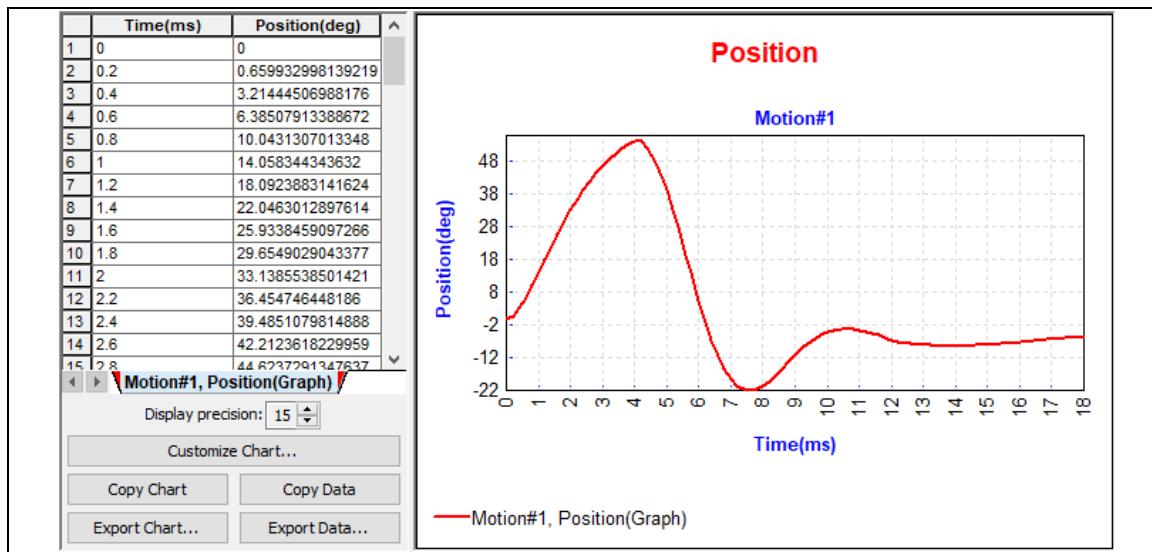
1. In the Results window, open the *Motion* page.



2. Using the mouse pointer, click anywhere inside the *Position* text box (i.e. 0 °).

3. Click the *Graph Selection* button, located at the top of the Results window.

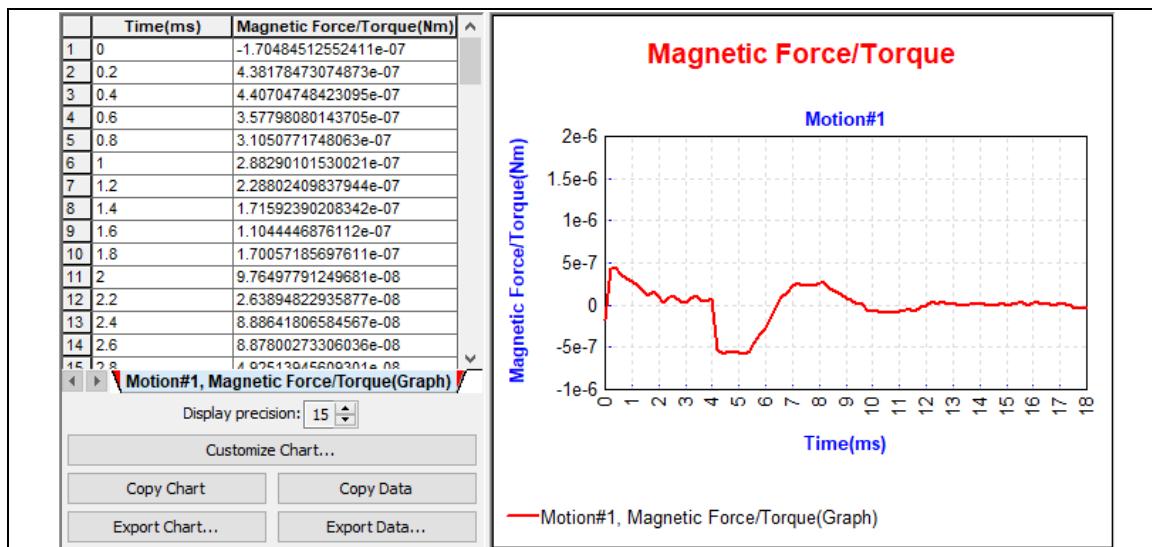
The graph should look like the illustration on the next page.



Note Each time that a graph is generated, a new window containing the chart automatically opens. To graph more results (as the following steps require) you must switch back to the Results window each time by clicking the Results tab located at the bottom of the window.

4. Using the mouse pointer, click anywhere inside the *Magnetic Force/Torque* text box (i.e. $-1.70484512552411e-07$ N.m).
5. Click the *Graph Selection* button.

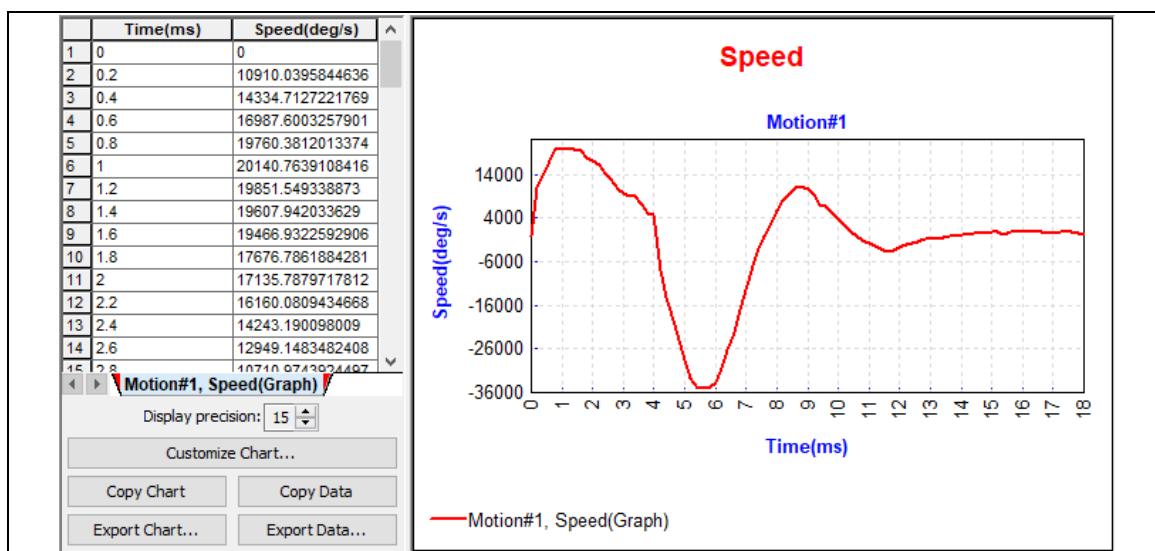
The graph should look like the illustration below.



6. Using the mouse pointer, click anywhere inside the *Speed* text box (i.e. 0 deg/s).

7. Click the *Graph Selection* button.

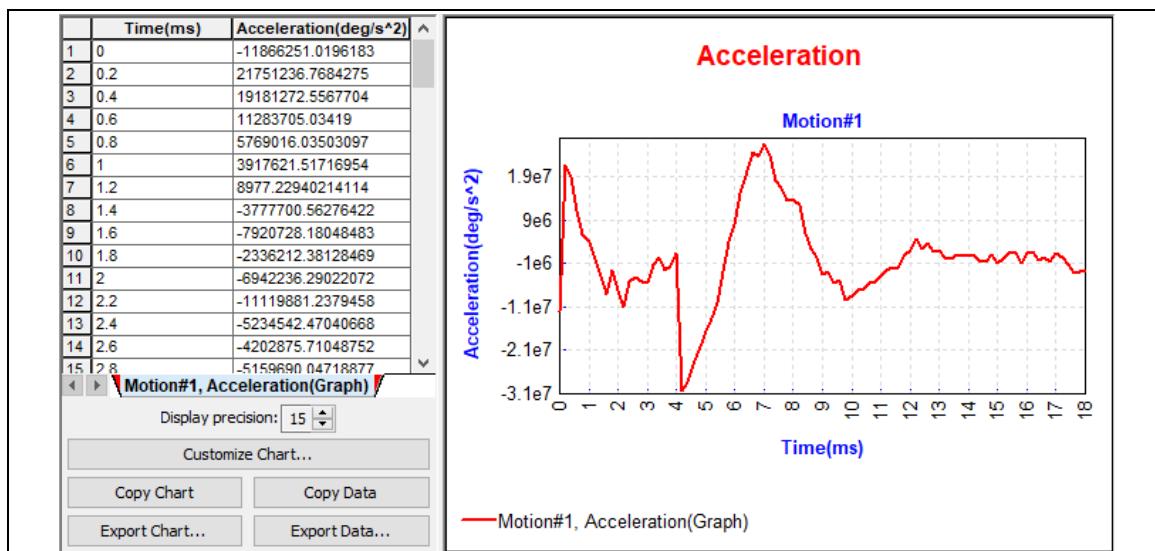
The graph should look like the illustration below.



8. Using the mouse pointer, click anywhere inside the *Acceleration* text box (i.e. $-11866251.0196183 \text{ } ^\circ/\text{s}^2$).

9. Click the *Graph Selection* button.

The graph should look like the illustration below.

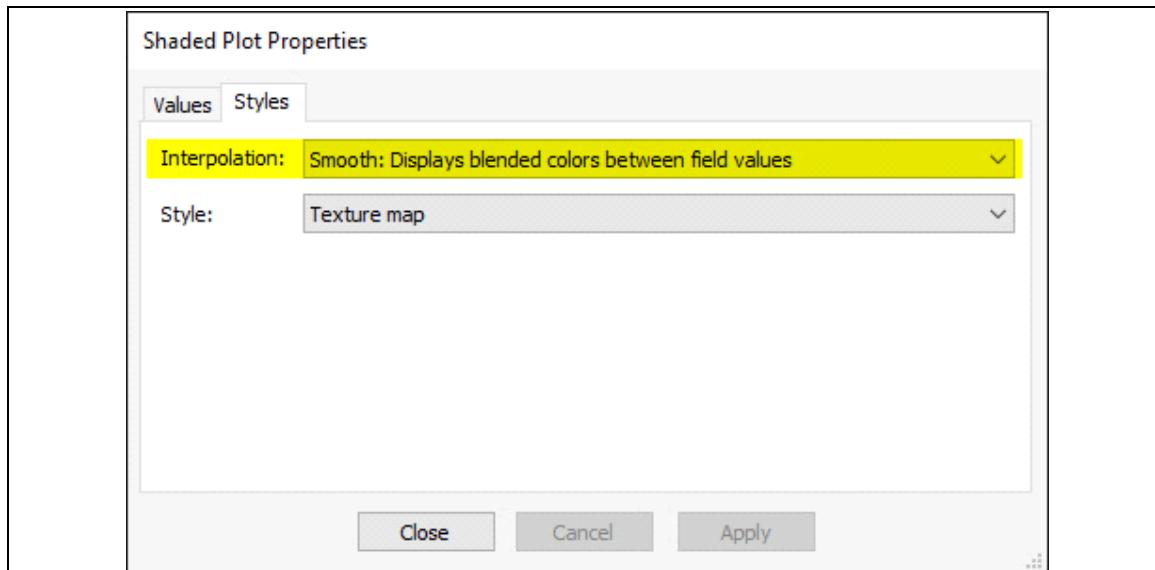


8.2 Set the color interpolation and style of the shaded plot

This procedure will set the default for shaded plots to smooth instead of discrete, which is the default.

1. On the View menu, click *Default Fields*.
2. On the Project Bar, select the *View* tab.
3. From the View tree, click *Shaded Plot*.
4. On the Edit menu, click *Properties*.

The Shaded Plot Properties page appears.



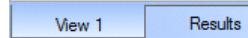
5. Click the Styles tab and then from the Interpolation drop-down menu, select "Smooth: Displays blended colors between field values".
6. Click OK.

8.3 View the shaded plot

The shaded plot displays shaded lines of the flux density. These shaded lines are the magnitude of the flux density. Hiding the Air box component allows us an unobstructed view of the mesh.

Note If the view is already zoomed-in around the rotor, you can skip steps 4 and 5.

1. Before viewing the shaded plot, switch back to the View window by clicking the View tab



located at the bottom of the window.

2. Select Air box from the Object page.

3. Right-click and then click *Visible*.

The component is hidden from view. A red “X” appears by the Air box component’s name in the Object page.

4. On the View menu, click *Preset Views*, and then *Negative Z Axis*.

5. On the View toolbar, click (Examine Model).

6. Hold down the CTRL key and the left mouse button to form a rectangular box around the Rotor and Remesh Region, including the part of the Stator that envelops these two components.

7. On the Project bar, select the Field tab.

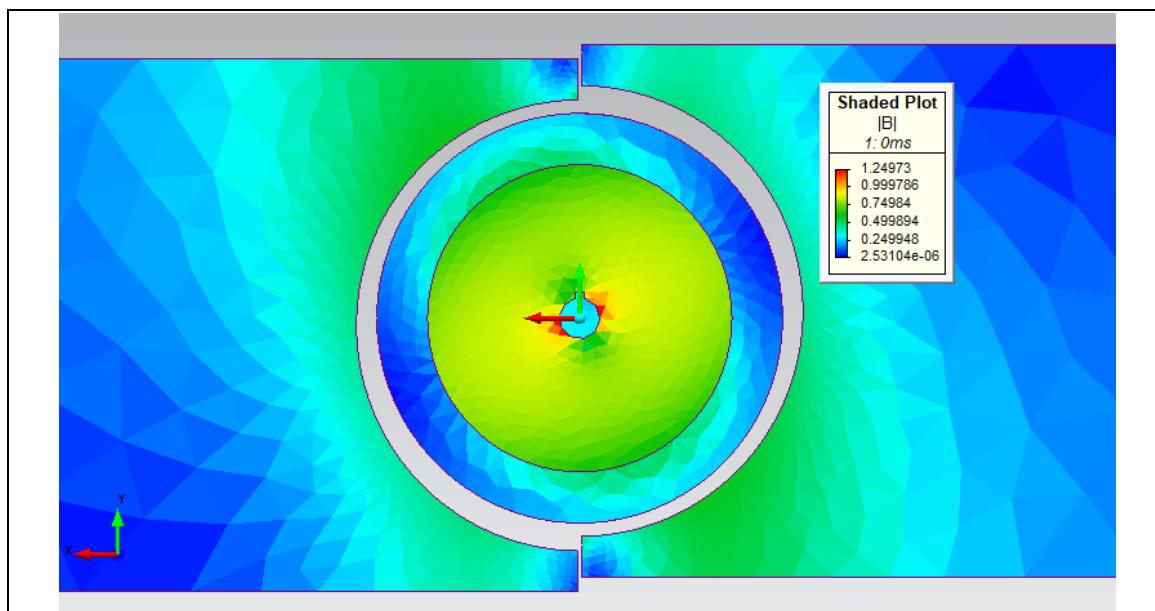
The Field page opens.

8. Click the shaded plot tab (at the bottom of the Field page).

9. In the *Fields to Display* list, select $|B|$.

10. At the bottom of the Field page, press Update View.

The shaded plot is displayed. (This may take a moment.)

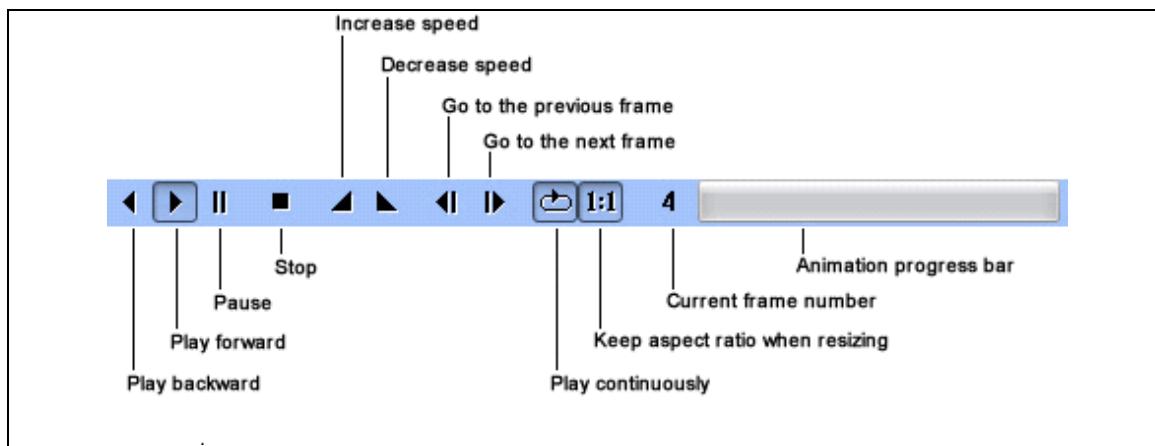


8.4 Animate the shaded plot

An animation is a series of “snapshots”, or frames, that are viewed as a moving image.

1. On the Field page, click  (Animate button).

Please wait a moment while the animation is created. When the animation is complete, an Animation window appears. The animation automatically plays in a loop. Use the Animation Control toolbar to control the playing of the animation.



Note The Animation Control toolbar may be hidden from view. If this is the case, maximize the Animation window by double-clicking on its title bar. The Animation Control toolbar should now be visible.

2. Click the Stop button  when you are finished viewing the animation.

8.5 Save the animation

1. On the File menu, click *Save*.
The Save As dialog box appears.
2. In the File Name text box, enter **PM Stepper Motor**.
The animation extension .ban is automatically added.
3. Click *Save*.
The animation is saved.
4. On the File menu, click *Close*.
The Animation window closes.

9 Summary

In this tutorial, you completed the steps in creating a model for a transient solution. The skills you learned include:

- Setting up the work environment by modifying initial settings and the viewing area.
- Building the geometric model using the Keyboard Input Bar.
- Setting up the problem -- this consists of making components and coils, and modifying the mesh.
- Making a motion component along with a Remesh Region component.
- Generating the transient field solution using Simcenter MAGNET's Transient 3D with Motion solver.
- Analyzing the results, which includes:
 - graphing the position, magnetic torque, velocity, and acceleration of the motion component
 - creating an animation of the shaded plot of $|B|$