



# Tutorials

**Infolytica Corporation**

June 18, 2003

We welcome your comments regarding Infolytica Corporation documents. Please send comments or corrections to the following addresses:

**email:** docs@infolytica.com

**fax:** Documentation Department  
514.849.4239

**post:** Documentation Department  
Infolytica Corporation  
300 Leo Pariseau, Suite 2222  
Montreal, Quebec H2X 4B3  
Canada

© Copyright 2003, Infolytica Corporation.  
Part number M6T111

All rights reserved. No part of this document may be reproduced, translated to another language, stored in a retrieval system, or transmitted in any form or by any means, electronic, photocopying, recording, or otherwise, without written permission from Infolytica Corporation.

The information in this document is subject to change without notice.

# Table of Contents

<b>Chapter 1</b>	<b>Introduction</b>	
	Introduction to the tutorials .....	7
	Description of the tutorials .....	8
<b>Chapter 2</b>	<b>2D Time-harmonic Tutorial: Cylindrical shield</b>	
	Open a new model .....	17
	Build the geometric model .....	18
	Modify the mesh .....	28
	Define boundary conditions .....	35
	Create the coil .....	38
	Edit the coil properties .....	39
	Set the source frequency .....	41
	Change the polynomial order .....	42
	Solve .....	43
	View the solution results .....	44
<b>Chapter 3</b>	<b>2D Transient Tutorial: Felix long cylinder</b>	
	Copy the basic model .....	52
	Create a circuit .....	53
	Edit the waveform properties .....	57
	Solve .....	61
	View the solution results .....	62
<b>Chapter 4</b>	<b>2D Tutorial: Spherical shield (basic model)</b>	
	Open a new model .....	71
	Build the geometric model .....	72
	Modify the mesh .....	81
	Define boundary conditions .....	89
	Create the coil .....	92

<b>Chapter 5</b>	<b>2D Magnetostatic tutorial: Spherical shield</b>	
	Copy the basic model	94
	Change the material of the sphere	95
	Edit the coil properties	96
	Solve	98
	View the solution results	99
<b>Chapter 6</b>	<b>2D Time-harmonic Tutorial: Spherical shield</b>	
	Copy the basic model	106
	Create a new material	107
	Change the material of the sphere	108
	Set the source frequency	109
	Solve	110
	View the solution results	111
<b>Chapter 7</b>	<b>3D Magnetostatic tutorial: Pot-core with a coil</b>	
	Open a new model	118
	Build the geometric model	119
	Create the excitation	133
	Add a boundary condition	140
	Refine the mesh	143
	Solve	151
	View the solution results	152
<b>Chapter 8</b>	<b>3D Time-harmonic tutorial: Bath plate</b>	
	Open a new model	159
	Build the geometric model	160
	Create the excitation	163
	Add a boundary condition	168
	Set the source frequency	171
	Solve	172
	View the solution results	173

<b>Chapter 9</b>	<b>3D Transient Tutorial: Felix short cylinder</b>	
	Open a new model . . . . .	180
	Build the geometric model . . . . .	181
	Modify the mesh . . . . .	190
	Define boundary conditions . . . . .	197
	Create the coil . . . . .	200
	Edit the coil properties . . . . .	201
	Set the source frequency . . . . .	202
	Change the polynomial order . . . . .	203
	Create a circuit . . . . .	204
	Edit the waveform properties . . . . .	208
	Solve . . . . .	213
	View the solution results . . . . .	214
 <b>Chapter 10</b>	 <b>Parameterization tutorial: C-core with a rotating block</b>	
	Open a new model . . . . .	220
	Build the geometric model . . . . .	221
	Add a boundary condition . . . . .	227
	Parameterize the rotating block . . . . .	230
	Modify the mesh . . . . .	235
	Solve . . . . .	239
	View the solution results . . . . .	240
	 Index . . . . .	 247



# Chapter 1

## Introduction

### Introduction to the tutorials

This book includes nine tutorials to help you increase your skills with MagNet. Each tutorial teaches you basic procedures that you can apply to your own models.

#### 2D Tutorials

##### Translational geometry

Time-harmonic solution: Cylindrical Shield

Transient solution: Felix long cylinder

##### Rotational geometry

Spherical Shield (basic model)

Magnetostatic solution: Spherical Shield

Time-harmonic solution: Spherical Shield

#### 3D Tutorials

Magnetostatic tutorial: Pot-core with a coil

Time-harmonic tutorial: Bath plate

Transient tutorial: Felix short cylinder

Parametrization tutorial: C-core with a rotating block

#### Additional information

The Getting Started Guide includes two “Quick Start” tutorials to introduce you to the basic MagNet concepts. More information on the procedures and concepts of model building with MagNet is found in the Help included with each package. The MagNet CD also includes important guides in PDF format (the Adobe Acrobat reader required to view these guides is also included on the CD).

## Description of the tutorials

The features showcased in each tutorial are outlined below.

### **Chapter 2**     **2D Translational Geometry** Cylindrical Shield: Time-harmonic solution

**Estimated time:** 30 minutes.

#### **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 power dissipated in the conductor
- Viewing the contour plot of the model
- Probing a field value

### **Chapter 3**     **2D Translational Geometry** Felix long cylinder: Transient solution

In this tutorial, the cylindrical shield model built in Chapter 2 is used as the basis for the Felix long cylinder problem.

**Estimated time:** 30 minutes.

#### **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 power of each conducting component
- Graphing the power across time



## **Chapter 4**      **2D Rotational Geometry** **Spherical Shield (basic model)**

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

In Chapters 5 and 6, the model is edited for magnetostatic and time-harmonic solutions.

**Estimated time:** 20 minutes.

### **Features you will learn**

- Drawing with the Keyboard Input bar
- Creating components in a rotational direction
- Rotating the display of a model
- Modifying the default mesh edge subdivisions
- Defining boundary conditions
- Creating a coil

## **Chapter 5**      **2D Rotational Geometry** **Magnetostatic solution**

In this tutorial, the spherical shield model built in Chapter 4 is edited for a magnetostatic solution.

**Estimated time:** 20 minutes.

### **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

## **Chapter 6**      **2D Rotational Geometry** **Time-harmonic solution**

In this tutorial, the spherical shield model built in Chapter 4 is edited for a time-harmonic solution.

**Estimated time:** 20 minutes

### **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 power dissipated in the conductor
- Probing a field value
- Viewing the contour plot of the model

## **Chapter 7**      **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.

**Estimated time:** 1 hour and 15 minutes.

### **Features you will learn**

- Distorting the shape of a component
- Building a coil
- Refining the mesh by changing the mesh edge subdivisions
- Viewing an arrow plot

## **Chapter 8**      **Time-harmonic tutorial:** **Bath plate**

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

**Estimated time:** 45 minutes.

### **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

## **Chapter 9**      **3D Translational Geometry** **Felix short cylinder: Transient solution**

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

**Estimated time:** 30 minutes.

### **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

## **Chapter 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.

**Estimated time:** 30 minutes.

### **Features you will learn**

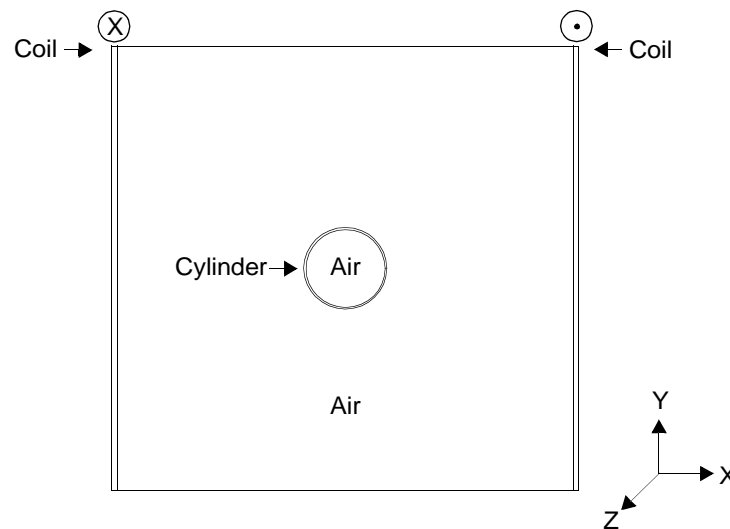
- Creating a user-defined parameter
- Rotating the geometric position of a component using parameterization
- Viewing instantiations of a model
- Viewing force and torque on a body
- Graphing the Z component of torque over multiple solutions
- Creating an animation of the shaded plot over multiple solutions

## Chapter 2

# 2D Time-harmonic Tutorial: Cylindrical shield

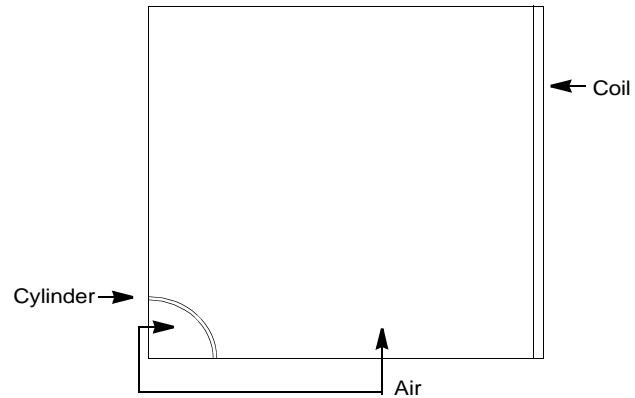
### Modeling plan

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

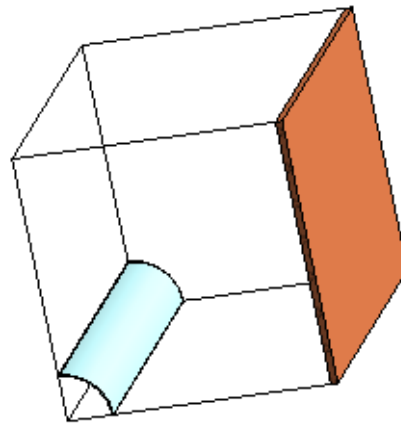


**Note** This model uses the same geometry as the magnetostatic Cylindrical Shield problem solved in the *Getting Started Guide*. If you have created the Cylindrical Shield model using the Getting Started Guide tutorial, you can save the model to a new name (Cylindrical Shield - Time-Harmonic) and change the material of the cylinder to Aluminum 6061 (see page 26 for the material properties). You can proceed to Step 7 on page 41.

Symmetry conditions allow for only one-quarter 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.

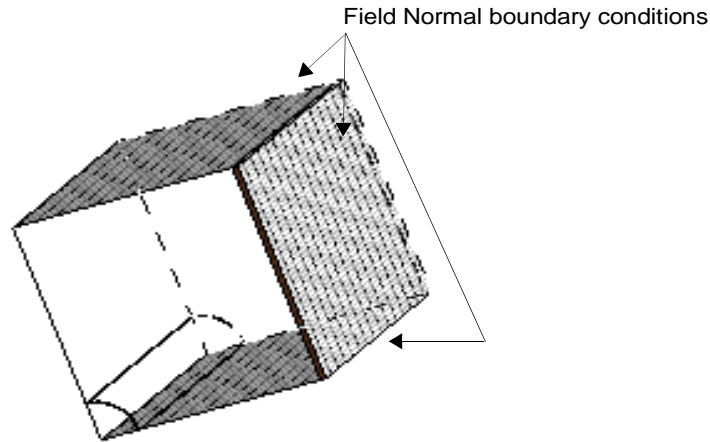


After drawing, the geometry is swept into components and a coil is created from one of the components.

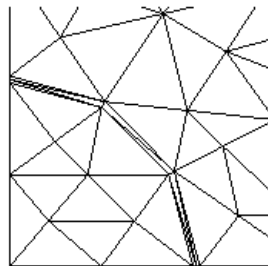


**Tip** The Field Normal boundary condition on the right surface of the air box (representing the outside of the coil) forces the coil flux outside to infinity. If the surface had the Flux Tangential boundary condition, the coil flux would be forced to return inside the coil (which would give incorrect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

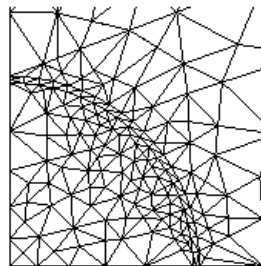
Boundary conditions are applied to indicate symmetry. The Field Normal boundary condition is applied to the top, bottom, and right surfaces of the air space. The default boundary condition, Flux Tangential, is automatically applied to the remaining surfaces.



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

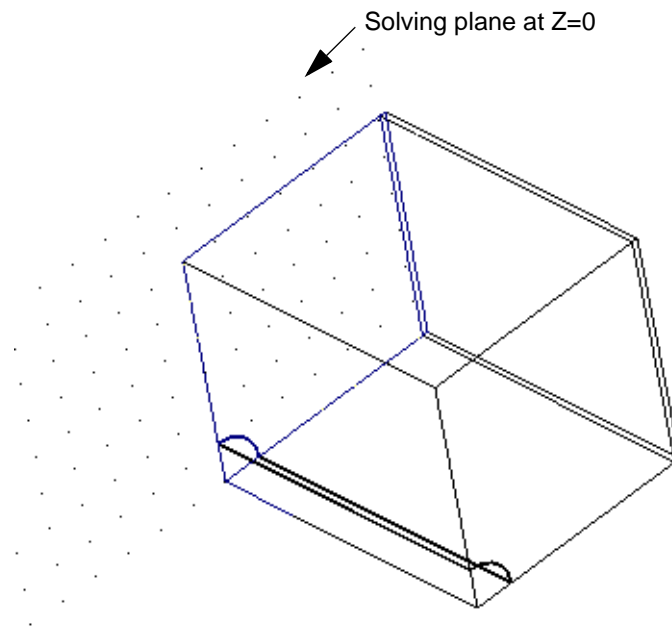


Default initial mesh



Modified initial mesh

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






## Step 1: Open a new model

---

**Note** This model uses the same geometry as the magnetostatic Cylindrical Shield problem solved in the *Getting Started Guide*. If you have created the Cylindrical Shield model using the Getting Started Guide tutorial, you can save the model to a new name (Cylindrical Shield - Time-Harmonic) and change the material of the cylinder to Aluminum 6061 (see page 26 for the material properties). You can then proceed to Step 7 on page 41.

### Open a new model

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

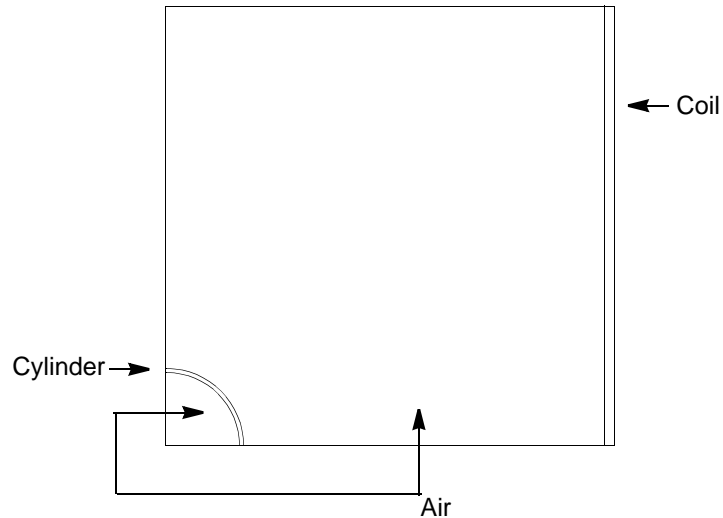
### Name the model

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

## Step 2: Build the geometric model


---

The geometry of the model is shown below.



### Set the drawing area

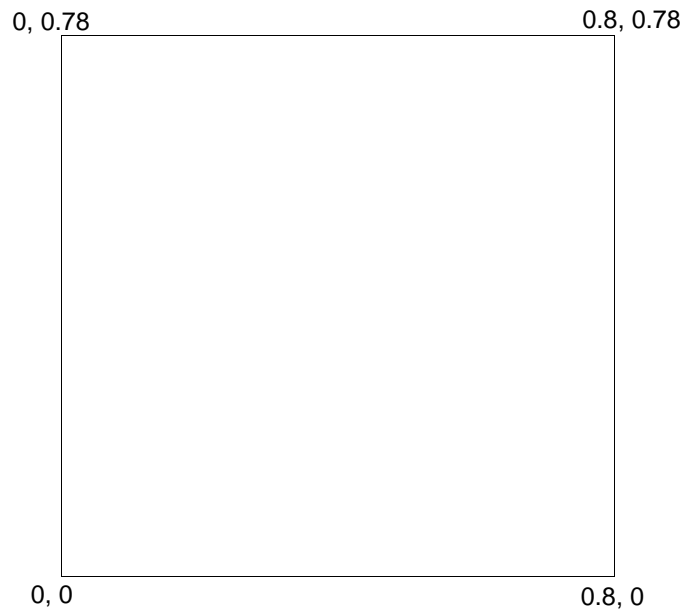
The model will be drawn using the Keyboard Input bar, so the construction grid is not needed and can be removed from the display. The drawing unit is meters (the MagNet default).

1. Turn off the display of the Construction Grid (if it is displayed).  
If the grid is visible, toggle Construction Grid on the View menu.
2. On the Zoom toolbar, click the Update Automatically tool .  
This option updates the display of the model to fit inside the View window.

## Create the air space component




### Draw the geometry of the air space

The geometry of the air space 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 the Line drawing tool .

4. 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

5. Press ESC to stop drawing.

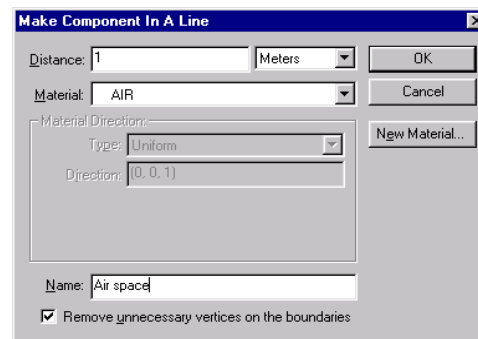
### Make the air space component

A component can now be made from the surface that you have drawn. The component is swept one unit length.

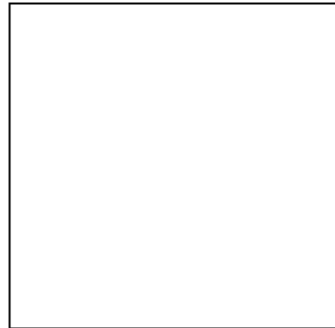
Components are created using the Make Component dialog box.

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 the Make Component in a Line tool .

The Make Component In A Line dialog box appears.



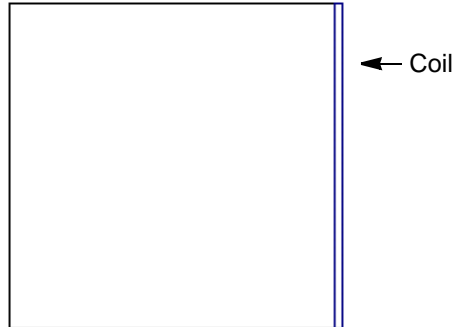
4. In the Distance box, enter 1.
5. In the Material drop down list, make sure AIR is selected.  
AIR is the default material.
6. In the Name box, enter **Air space**.
7. Click OK to accept the settings.  
The component is created.




8. On the File menu, click Save.

## Create the coil component

The geometry of the coil component is shown below.



### Draw the coil component

1. On the Draw toolbar, click the Line drawing tool .
2. 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
-----------------	--------	-------------

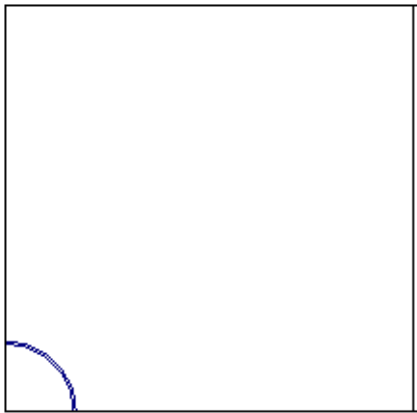
3. Press ESC.

### 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 the Make Component in a Line tool .
4. In the Material drop down list, select **Copper: 5.77e7 Siemens/meter**.
5. In the Name box, enter **Coil component**.
6. Click OK to accept the settings.
7. On the File menu, click Save.

### Create the cylinder component

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



## Draw the cylinder


1. On the Model toolbar, click the Arc drawing tool .
2. 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	0.1317, 0	Press ENTER
End coordinates	0, 0.1317	Press ENTER

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

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

4. On the Draw toolbar, click the Line drawing tool .
5. In the Keyboard Input bar, enter the following coordinates to draw the lines of the cylinder.

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


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

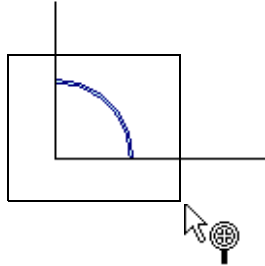
Start coordinates	0.1317, 0	Press ENTER
End coordinates	0.1365, 0	Press ENTER

8. Press ESC.

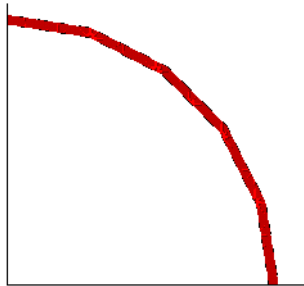



**Make the component of the cylinder**

1. On the Zoom toolbar, click the Zoom In tool .
2. Drag the mouse pointer 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.

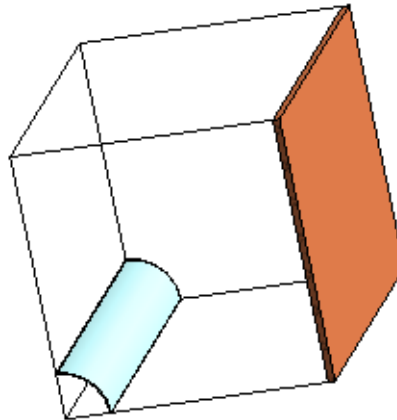


6. On the Model toolbar, click the Make Component in a Line tool .
7. In the Name box, enter **Cylinder**.
8. Click New Material.  
For this problem, you will have to create a new material in your material database.
9. On the General page, enter the following data:
  - Name: **Aluminum 6061**
  - Display color: *Click Set Color and select an appropriate color*
  - Description: *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**
  - Relative Permeability = **1**
  - Coercivity *Amps/m* = **0**
  - Conductivity *Siemens/m* = **2.538e7**
13. Once you have entered all the values, click Finish in the Confirmation page to create the new material.
14. From the Material drop down list, select **Aluminum 6061**.
15. Click Ok to accept the settings.
16. On the File menu, click Save.

**Rotate the display of the model**

1. On the Zoom toolbar, click the View All tool .
2. On the Preset View toolbar, click the Dynamic Rotation tool .
3. Click the mouse pointer on the model.
4. 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.
5. Release the mouse button.

The display is rotated about the center of the model. The model is displayed as a wireframe during the rotation.



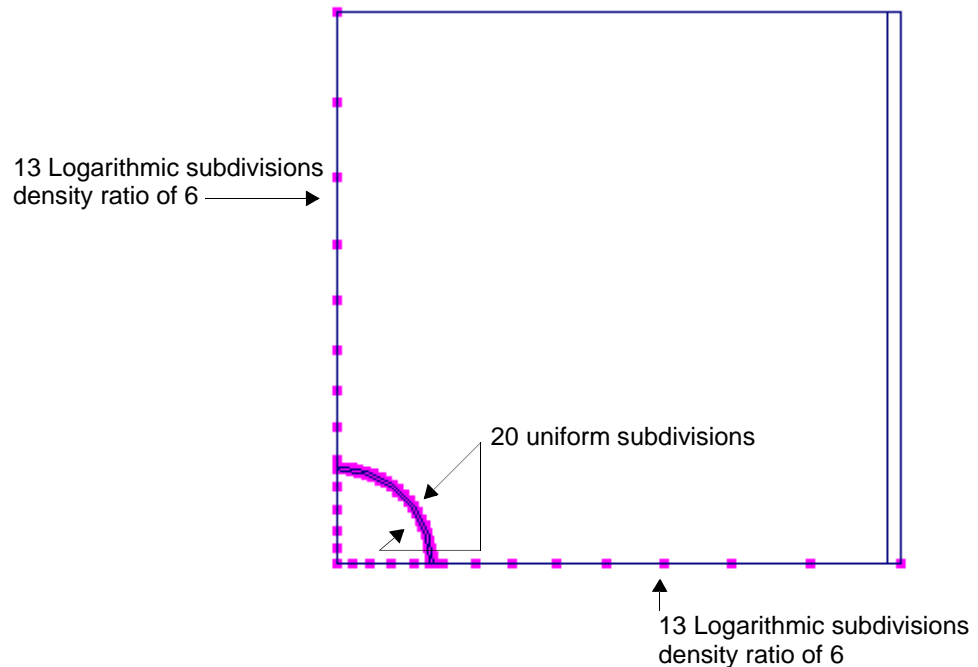
## Step 3: Modify the mesh

---

In the 2D finite element method of analysis, the model is divided into a mesh of triangular-shaped 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 small elements.


One method of increasing mesh density is to subdivide component edges into segments. The number of edge segments corresponds to the number of elements along the edge. Note that the subdivisions propagate throughout the entire model in the sweep direction.

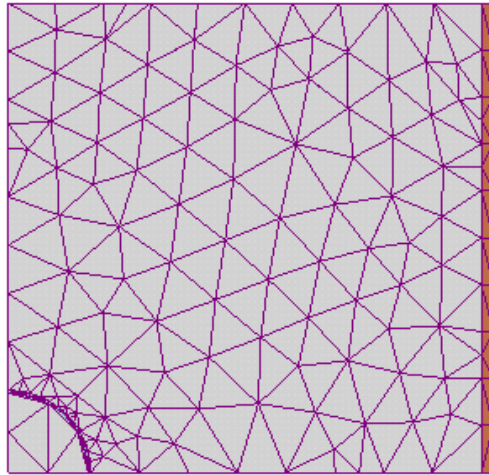
The edge subdivisions that will be modified in this model are shown in the diagram below.



## View the initial mesh

Before changing the subdivisions, the default initial mesh can be viewed.

1. On the View menu, click Initial 2D Mesh.  
The initial mesh appears in the View window. The initial mesh is displayed at the XY plane,  $Z=0$ .
2. On the Preset View toolbar, select  (positive Z axis).  
The mesh should look like the following diagram.




## Modify the edge subdivisions

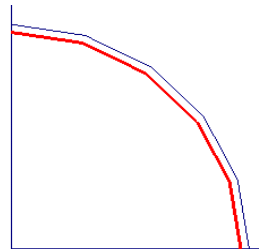
Subdivisions can be applied to an edge using either a logarithmic or uniform (linear) scale. Note that subdivisions are edited on the model, not directly on the mesh. Subdivisions are applied to the model using the Mesh toolbar. Please see the Help for instructions on displaying toolbars.




Mesh toolbar: Floating position


### Subdivide the edges of the cylinder

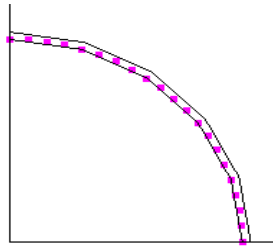
1. On the View menu, click Solid Model.
2. Use the Zoom In tool  to enlarge the area around the cylinder.
3. On the Edit menu, click Select Component Edges.
4. Click the mouse pointer on one of the arcs of the cylinder.  
The arc is highlighted.




5. On the Mesh toolbar, in the Number of Subdivisions text box (the leftmost text box)  , enter **20**.

**Tip** To improve the display of arcs, you can decrease the smoothness angle using the `SetCurveSmoothnessAngle` scripting command. Please see the Help for more information.

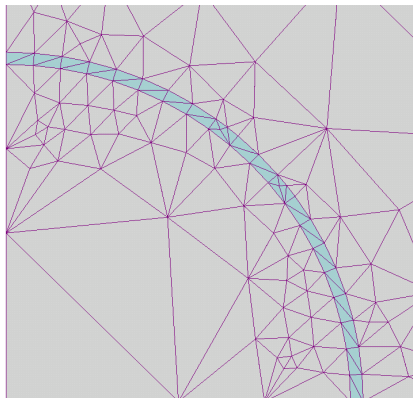
6. On the Mesh toolbar, select the Uniform Subdivision tool . 20 uniform subdivisions are assigned to the edge.




7. Click the mouse pointer on the other arc of the cylinder.
8. On the Mesh toolbar, select the Uniform Subdivision tool . 20 uniform subdivisions are assigned to the edge.

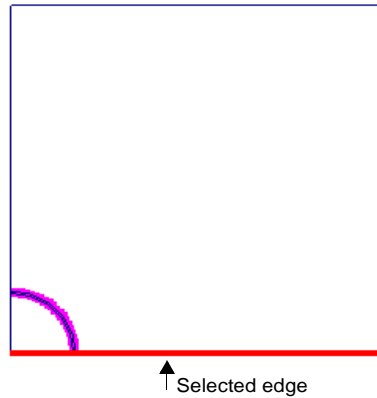
### View the changes to the mesh


- On the View menu, click Initial 2D Mesh.  
The mesh updates (this may take a moment).  
The mesh should look like the following diagram.



### Subdivide the edges of the air space

1. On the View menu, click Solid Model.
2. On the Zoom toolbar, click the View All tool .
3. Click the mouse pointer on the edge of the air space (as shown in the following diagram).




4. On the Mesh toolbar, in the Number of Subdivisions text box (the leftmost text box) , enter **13**.
5. In the Subdivision Density Factor text box (the rightmost text box), enter **6**. The toolbar should look like the diagram below.




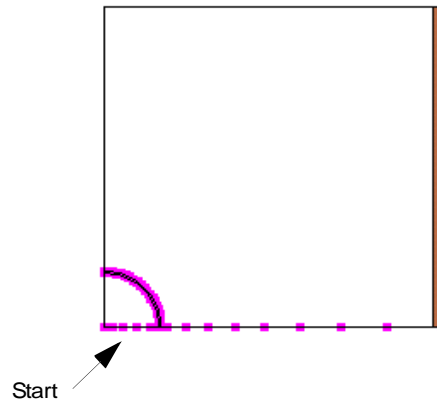
The density factor determines the length of the longest segment in relation to the shortest segment. For example, a density factor of 6 means that the longest segment is 6 times longer than the shortest segment.



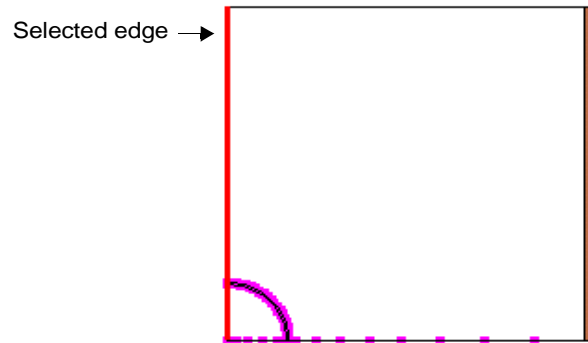
6. On the Mesh toolbar, click the Logarithmic: Concentrated at Start tool .


This setting places more subdivisions at the start of the edge than at the end of the edge. The subdivisions should look like the following diagram.

**Tip** If the subdivisions are concentrated at the other end of the edge, click the Logarithmic: Concentrated at End tool .




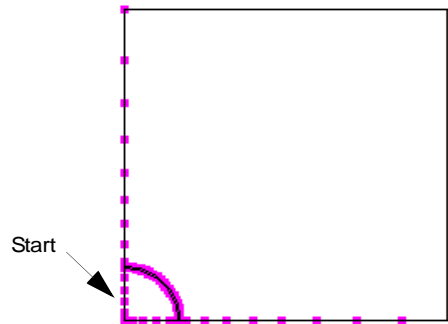
7. Click the mouse pointer on the edge of the air space as shown below.



8. On the Mesh toolbar, click the Logarithmic: Concentrated at End tool .


This setting places more subdivisions at the end of the edge than at the start of the edge. The subdivisions should look like the following diagram.

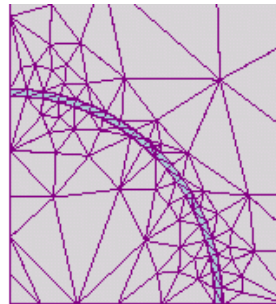
**Tip** If the subdivisions are concentrated at the end of the edge, click the Logarithmic: Concentrated at Start tool .



9. On the File menu, click Save.

### View the changes to the mesh

1. On the View menu, click Initial 2D Mesh.  
The mesh updates (this may take a moment).
2. Use the Zoom In tool  to enlarge the area around the cylinder.  
The mesh should look like the following diagram.



## Step 4: Define boundary conditions



---

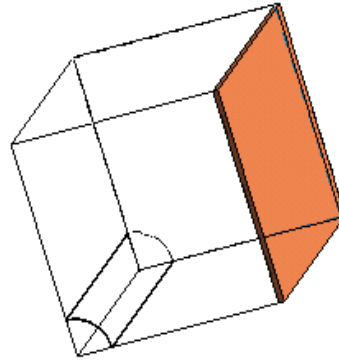
The Field Normal boundary condition is applied to three surfaces of the air space: the top, bottom, and right surfaces. The default boundary condition, Flux Tangential, is automatically applied to the remaining surfaces.

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 (along side) the boundary.

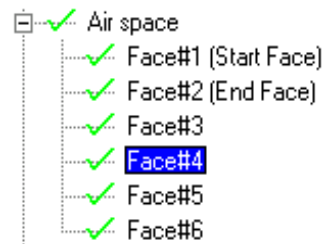
### Apply the Field Normal boundary condition

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

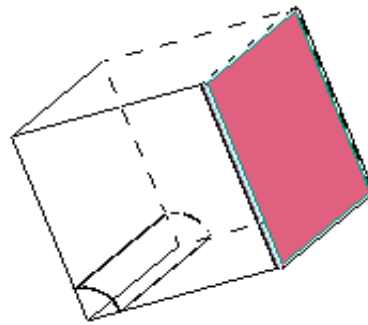
1. On the View menu, click Solid Model.
2. On the Zoom toolbar, click the View All tool .
3. On the Preset View toolbar, click the Dynamic Rotation tool .
4. Rotate the model to a 3D view (similar to the diagram below). This rotation will display the surfaces to which the Field Normal boundary condition will be applied.



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

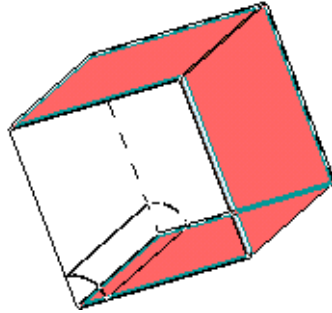



The surface is selected in the View window.

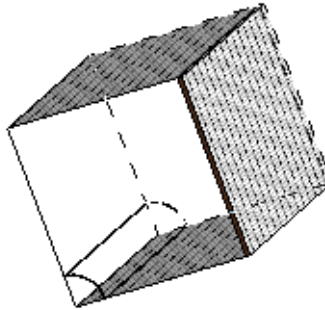


**Tip** You can select multiple objects by holding down the SHIFT or CTRL keys while clicking on additional objects. If you press CTRL while clicking on objects, you can click on an object a second time to de-select it.

7. While holding down the CTRL key on your computer keyboard, also select Air space, Face#3 and Face#5. The three surfaces are selected in the View window.



8. On the Boundary Condition toolbar, click Field Normal .
9. The Field Normal boundary condition is applied to the selected surfaces.



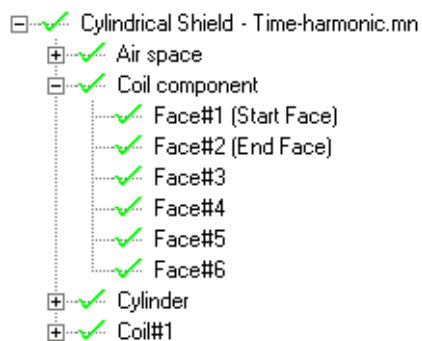
## Step 5: Create the coil

---

The coil is created from the coil component. After the coil is created, the default properties are edited.

1. On the Object page, select Coil component, Face#1 (Start Face).
2. On the Model menu, click Make Simple Coil.

The coil is listed in the Object page as Coil#1.

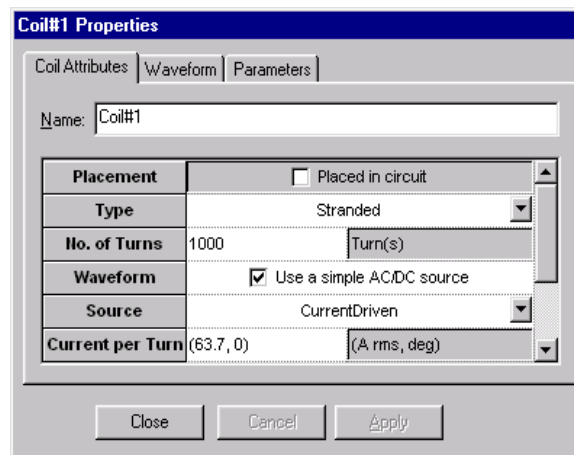


3. On the File menu, click Save.

## Step 6: Edit the coil properties

---

1. On the Project bar, click 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. In the No.Of Turns box, enter **1000**, then press Enter.
5. In the Current Per Turn box, enter **(63.7, 0)**, then press Enter.  
The current is 63.7, the phase is 0.



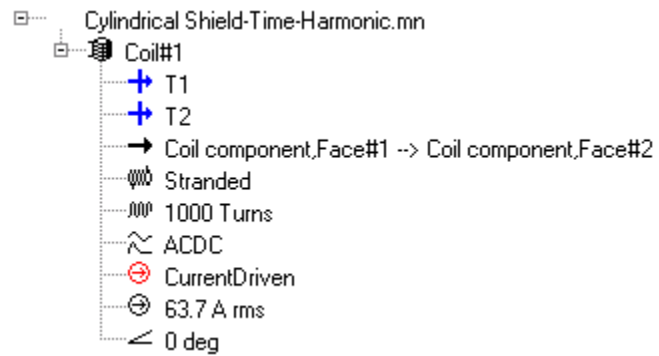
The image shows the 'Coil#1 Properties' dialog box with the 'Parameters' tab selected. The 'Name' field is 'Coil#1'. The 'Placement' section has a checkbox for 'Placed in circuit' which is unchecked. The 'Type' is 'Stranded'. The 'No. of Turns' is '1000' with the unit 'Turn(s)'. The 'Waveform' section has a checked box for 'Use a simple AC/DC source'. The 'Source' is 'CurrentDriven'. The 'Current per Turn' is '(63.7, 0)' with the unit '(A rms, deg)'. At the bottom are 'Close', 'Cancel', and 'Apply' buttons.

Coil#1 Properties	
Coil Attributes   Waveform   Parameters	
Name:	Coil#1
Placement	<input type="checkbox"/> Placed in circuit
Type	Stranded
No. of Turns	1000 Turn(s)
Waveform	<input checked="" type="checkbox"/> Use a simple AC/DC source
Source	CurrentDriven
Current per Turn	(63.7, 0) (A rms, deg)

Close Cancel Apply

**6.** Click OK.

The coil page is automatically updated.

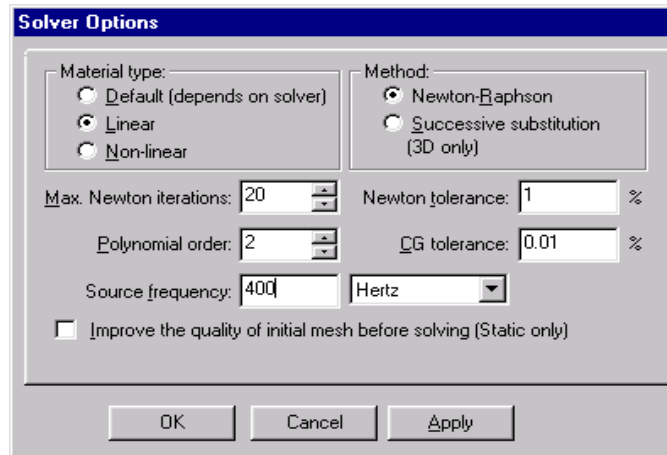
**7.** On the File menu, click Save.



## Step 7: Set the source frequency

---

1. On the Solve menu, click Set Solver Options.  
The Set Solver Options dialog appears.



**Tip** If the Linear option is not set, the solver will automatically detect the presence of linear materials.

2. Select the Linear option.
3. In the Source Frequency box, enter **400**.  
The default unit is Hertz.
4. Click OK.

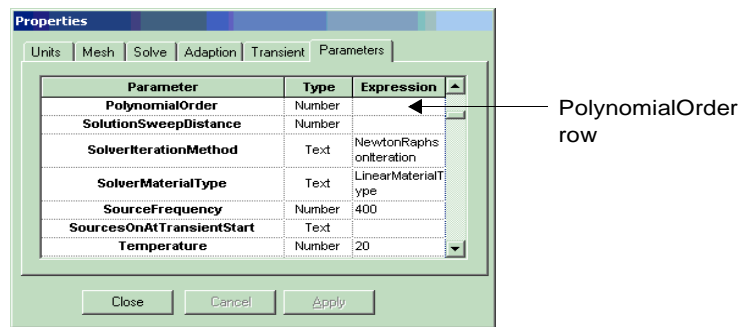
## Step 8: Change the polynomial order

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.

The polynomial order is set in the Model property page.

### To change the polynomial order

1. On the Object page, select Cylindrical Shield - Time harmonic.
2. On the Edit menu, click Properties.  
The Model property page appears.
3. Click the Parameters tab.



4. Scroll down the list of parameters until the PolynomialOrder row appears.
5. In the Expression column, enter 2, then press Tab.

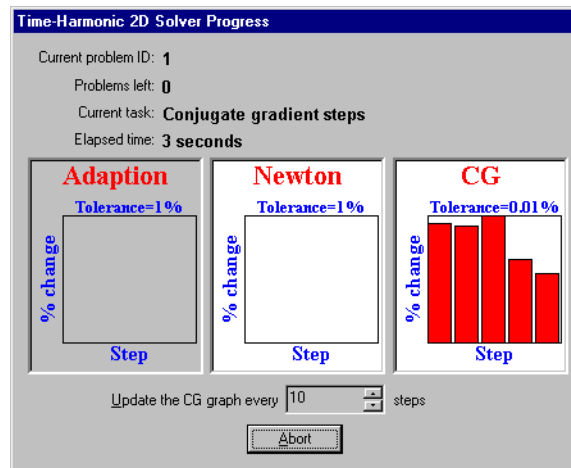
PolynomialOrder	Number	2
-----------------	--------	---

6. Click OK.
7. On the File menu, click Save.

## Step 9: Solve

---

- On the Solve menu, click Time-harmonic 2D.  
The Time-harmonic 2D Solver Progress dialog appears.



The cylindrical shield takes less than 10 seconds to solve (solving time may vary according to computer). The Solver Progress dialog automatically exits when the solution is complete.

## Step 10: View the solution results


---

The following results will be reviewed in this section:

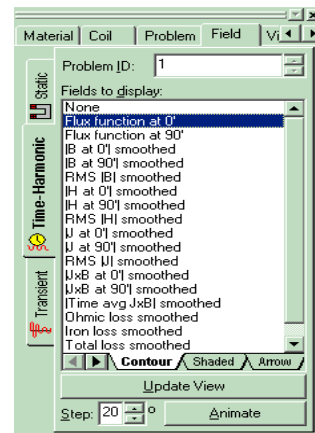
- The contour plot
- The time-averaged Ohmic power loss in the conductor
- The magnitude of B at (0, 0)

### View the contour plot

The contour plot displays contour lines of the magnetic flux function. These contour lines are the magnetic flux lines (lines that are everywhere parallel to the flux density vector).


1. On the Project bar, select the Field tab. (Use the arrow on the Project bar  to scroll the pages if necessary.)

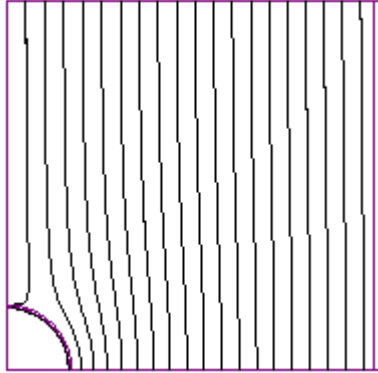
The Field page opens.



2. Click the Contour plot tab (at the bottom of the Field page).
3. In the Fields To Display list, select Flux Function at 0°.
4. At the bottom of the Field page, press Update View.

The contour plot is displayed. (This may take a moment.)

**Tip** If the View window is not displaying the entire contour plot, click the View All tool  on the Zoom toolbar.



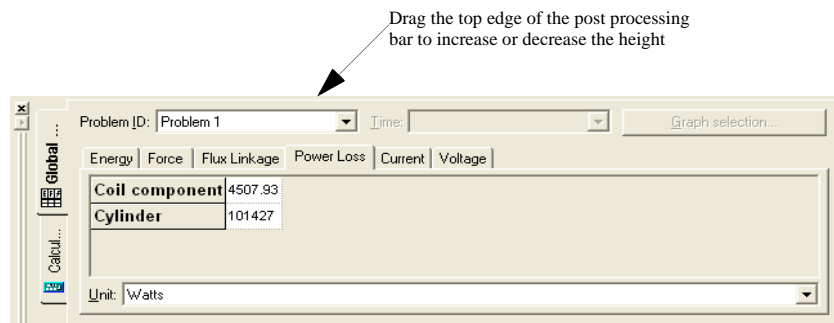
## View the time-averaged Ohmic power

The Post Processing bar is automatically displayed at the bottom of the Main window when the solution is complete.

1. On the Post Processing bar, click the Global Quantities tab.
2. Click the Power Loss tab.

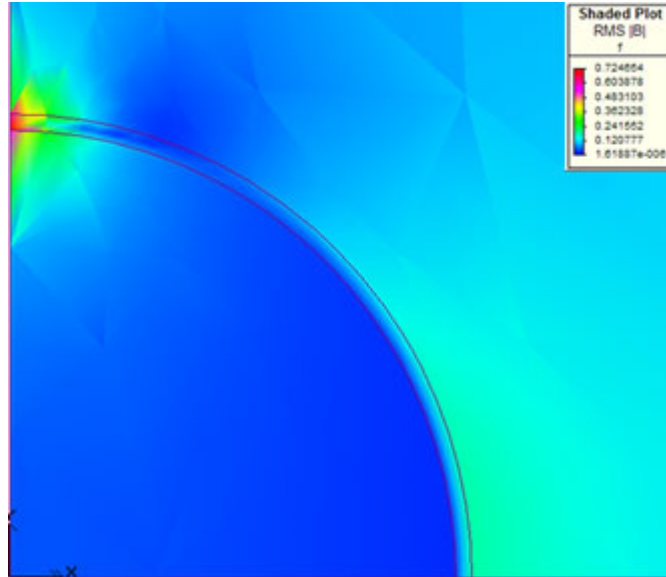
The Power Loss page displays the time-averaged Ohmic power loss in each conducting component in the model.

**Tip** Adjust the height of the Post Processing bar by placing the cursor on its top edge, then while keeping the left mouse button pressed, dragging the bar up or down.



## View the shaded plot of $|B|$

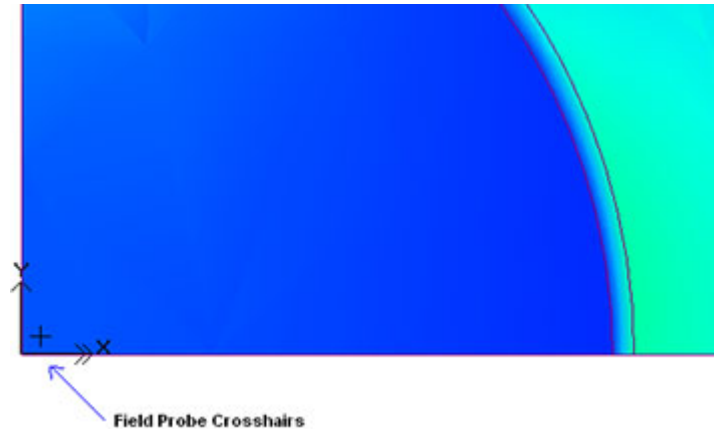
1. On the Project bar, select the Field tab.
2. Select the Contour page.
3. In the Fields To Display list, make sure that **None** is selected.
4. Select the Shaded page.
5. In the Fields To Display list, select **RMS  $|B|$** .
6. Select Update View.



The shaded plot on the surface of the model is displayed. (This may take a moment.) A color legend is displayed beside the shaded plot

## Probe the magnitude of 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 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.



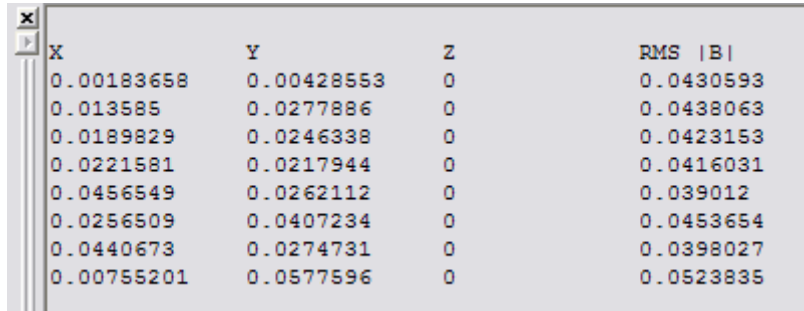
### To probe for field values using the mouse

1. On the Tools menu, click Field Probe.  
A checkmark indicates that it is enabled.
2. Move the mouse (crosshairs) over the solution near (0,0).  
The field value, for that specified location on the solution, is displayed in the Status Bar.

RMS |B| = 0.0426868      m    X: N/A    Y: N/A    Xg: 0.007269    Yg: 0.000476    Zg: 0

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

The Text Output Bar automatically opens (if it isn't already), and the field value and its location (x, y, and z coordinates) on the solution is displayed.



X	Y	Z	RMS  B
0.00183658	0.00428553	0	0.0430593
0.013585	0.0277886	0	0.0438063
0.0189829	0.0246338	0	0.0423153
0.0221581	0.0217944	0	0.0416031
0.0456549	0.0262112	0	0.039012
0.0256509	0.0407234	0	0.0453654
0.0440673	0.0274731	0	0.0398027
0.00755201	0.0577596	0	0.0523835

The example above shows several field values (under the heading |B|) and their location for every area that was clicked upon.

## Save the model

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

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



## 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:

- Creating a new material
- Editing the properties of a coil
- Setting the source frequency
- Setting the linear solving option
- Changing the polynomial order of the model
- Viewing the Contour plot of the model
- Viewing the time-averaged Ohmic power loss in the conductor
- Viewing the Shaded plot of the model
- Probing a field value using the Field Probe feature



## Chapter 3

# 2D Transient Tutorial: Felix long cylinder

In this tutorial, the cylindrical shield modeled in Chapter 2 is used as the basis for the Felix long cylinder. This model consists of a hollow aluminum cylinder placed in a uniform magnetic field. The magnetic field is perpendicular to the axis of the cylinder and decays exponentially with time.

The following properties of the model are edited:

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

After solving, the instantaneous power of the cylinder at the last time step is viewed and graphed. The contour plot at the last time step is also viewed and animated.

## Step 1: Copy the basic model

---

**Note** If you haven't created the Cylindrical Shield model featured in Chapter 2, please create the model before proceeding with the instructions below (follow Steps 1 through 3 in Chapter 2).

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.
4. Click Open.
5. On the File menu, click Save As.
6. In the Save As dialog box, enter **Felix long cylinder** as the name of the model.
7. Choose the drive and directory where you want to place the model.
8. Click Save.

## Step 2: 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.



Circuit Components toolbar



Circuit Move toolbar

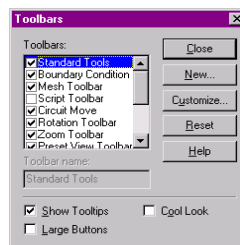


Circuit Alignment toolbar

### Display the Circuit toolbars

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

1. On the Tools menu, click Toolbars.  
The Toolbars dialog is displayed.

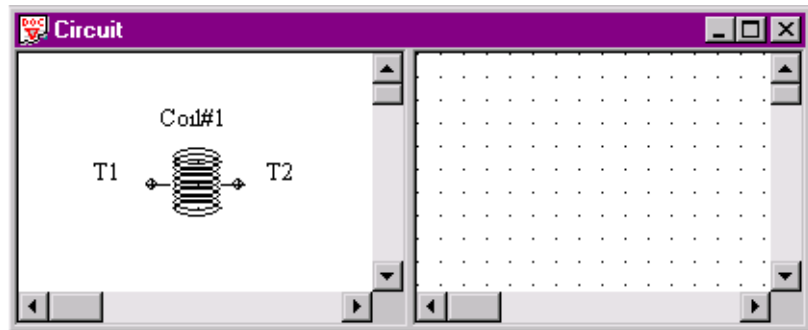


2. In the Toolbars list, select the following toolbars:
  - Circuit Move
  - Circuit Alignment
  - Circuit Components
3. On the Toolbars dialog, click Close.  
The toolbars are displayed.


## Create the circuit

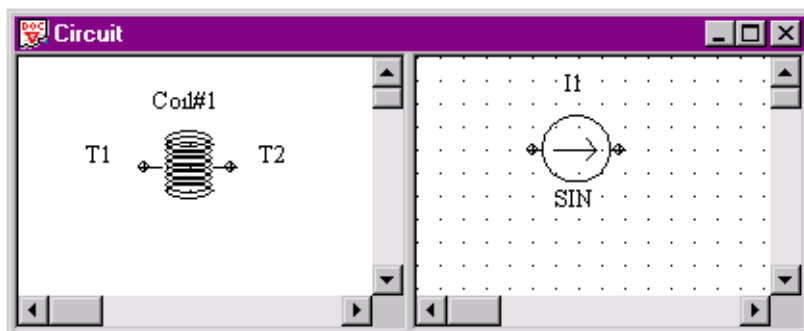
1. On the Circuit menu, click New Circuit window.


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

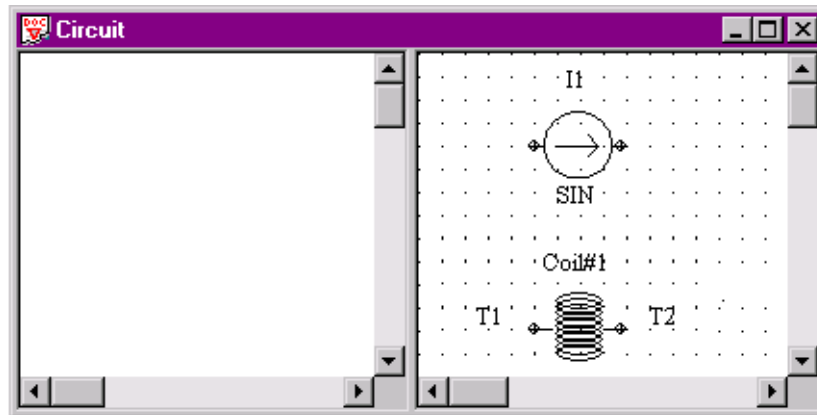


**Tip** The Circuit tools are also available on the Circuit menu. For example, to add a current source to the circuit, select Current Source on the Circuit menu.



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 highlighted 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. On the Selection toolbar, click the Select tool .
5. Select the coil in the left pane of the window, and then drag the coil to the right pane.
6. If necessary, re-size the window by dragging on an edge of the window.

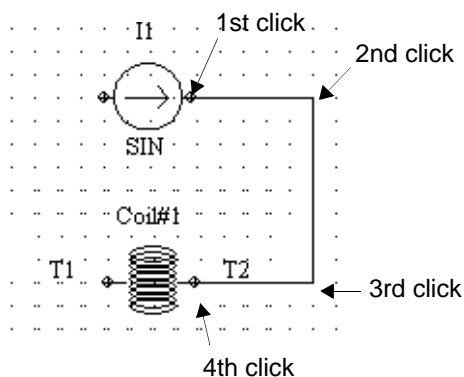


**Tip** You can select multiple objects by holding down the SHIFT or CTRL keys while clicking on additional objects. If you press CTRL while clicking on objects, you can click on an object a second time to de-select it.

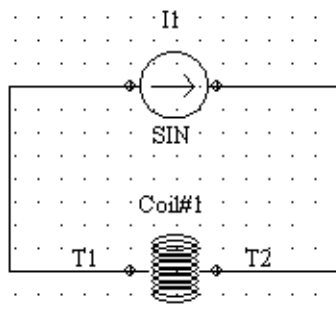
7. Select the coil and the current source with the mouse pointer. Press the SHIFT key on your keyboard while selecting the components.
8. On the Circuit Align toolbar, click Align at Center . The components are aligned at the position of the last selected component.
9. On the Circuit Components toolbar, 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.

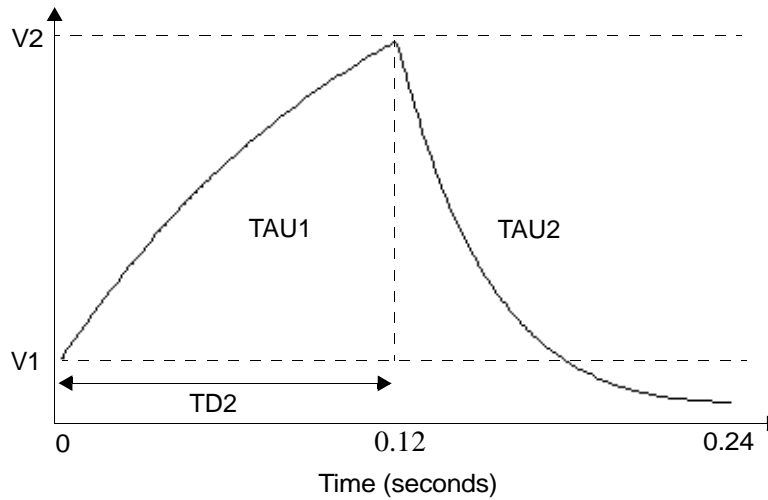


13. On the File menu, click Save.



## Step 3: Edit the waveform properties

The exponential waveform of the source looks like the following diagram:



The properties of the waveform are as follows:

<b>V1</b>	0	amperes
<b>V2</b>	98.19	seconds
<b>TD1</b>	0	seconds
<b>TAU1</b>	0.12	seconds
<b>TD2</b>	0.12	seconds
<b>TAU2</b>	0.0397	seconds
<b>START</b>	0.12	seconds
<b>STOP</b>	0.24	seconds
<b>STEP</b>	0.005	seconds

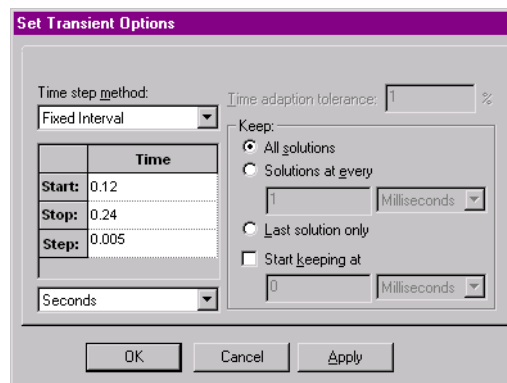
## Set the start, stop and step times

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.


**Note** The transient solver assumes the source values before the start time are equal to the values at the start time **only if** the parameter SourcesOnAtTransientStart is set to Yes.

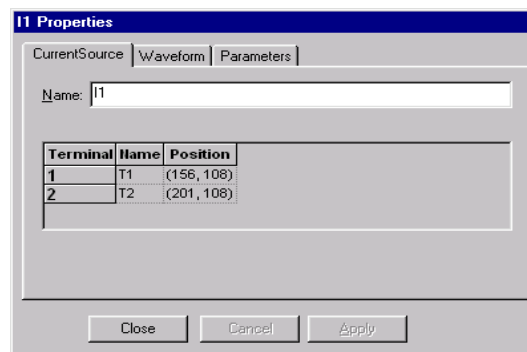
1. From the Object page, select the model (i.e., Felix Long Cylinder.mn).
2. On the Edit menu, click Properties.
3. Select the Parameters page.
4. Scroll down to the parameter **SourcesOnAtTransientStart**, type **Yes** in the Expression column, and then press Tab.
5. Click OK.
6. On the Solve menu, click Set Transient Options.  
The Set Transient Options dialog appears.



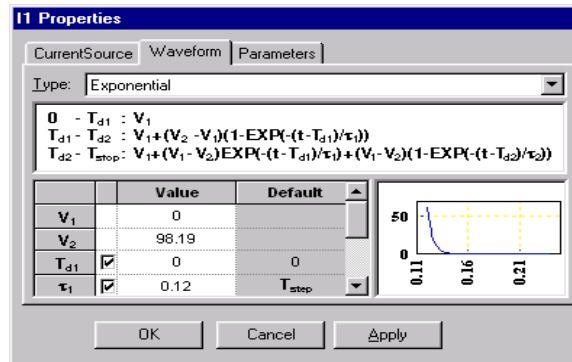
7. Make sure that *Fixed Interval* is selected as the Time Step method and *Seconds* as the unit for time, and then make the following modifications for Time:
  - Start = **0.12** Seconds
  - Stop = **0.24** Seconds
  - Step = **0.005** Seconds
8. Click OK.

## Edit the waveform

1. On the Selection toolbar, click the Select tool .
2. In the right pane of the Circuit window, right-click the current source.
3. On the pop-up menu, click Properties.  
The Current Source property page appears.



4. On the Current Source property page, click the Waveform tab.  
The Waveform property page is displayed.



5. In the Waveform page, make the following modifications:

- In the Type drop down list, select **Exponential**.
- In the V<sub>1</sub> text box, type **0**.
- In the V<sub>2</sub> text box, type **98.19**.
- For T<sub>d1</sub>, click the check box, and then type **0**.
- For T<sub>d2</sub>, click the check box, and then type **0.12**.
- For T<sub>stop</sub>, click the check box, and then type **0.0397**.

6. Click Apply.

The waveform is displayed in the property page. Note that the waveform is displayed from the start time of 0.12 seconds.

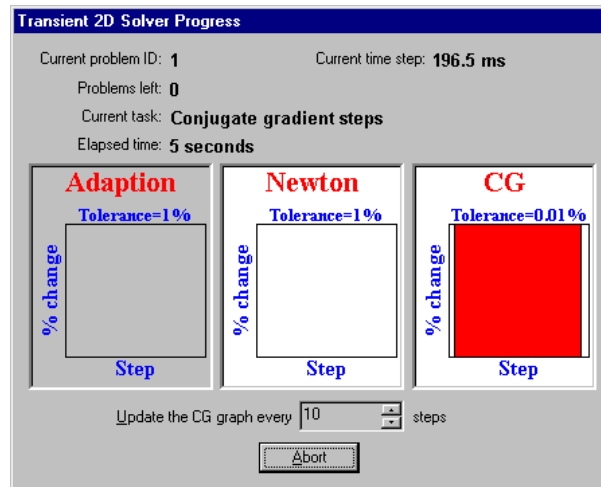
7. Click Close.

8. On the File menu, click Save.

## Step 4: Solve

---

- On the Solve menu, click Transient 2D.  
The Transient 2D Solver Progress dialog appears.



The cylindrical shield takes less than 10 seconds to solve (solving time may vary according to computer). The Solver Progress dialog automatically exits when the solution is complete.


## Step 5: View the solution results

The following results will be reviewed in this section:

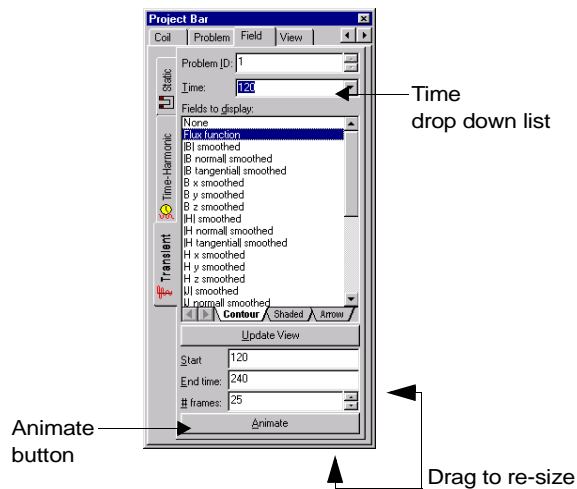
- The contour plot at the first time step
- An animation of the contour plot across time
- The instantaneous power at the last time step of each conducting component
- A graph of the instantaneous power across time

### View the contour plot

The contour plot displays contour lines of the magnetic flux function. These contour lines are the magnetic flux lines (lines that are everywhere parallel to the flux density vector).

1. On the Project bar, select the Field tab. (Use the arrow on the Project bar  to scroll the pages if necessary.)

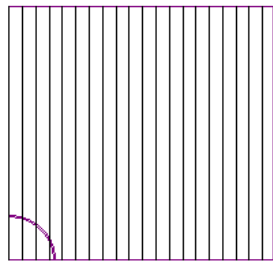
The Field page opens. You can re-size the Field page by dragging on the edges shown below.



2. Click the Contour plot tab (at the bottom of the Field page).


**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.

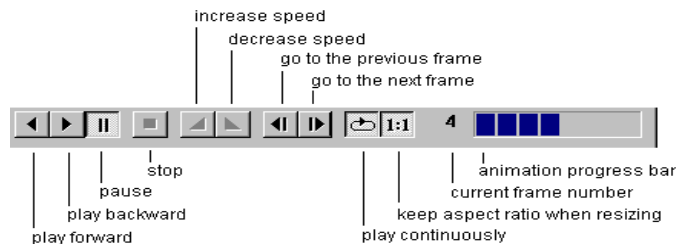
3. In the Fields To Display list, select Flux Function.
4. In the Time drop down list, make sure 120 is selected (the first time step).  
The default time unit is milliseconds.
5. At the bottom of the Field page, press Update View.  
The contour plot is displayed. (This may take a moment.)




## Animate the contour plot

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

1. On the Field page, select the 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.

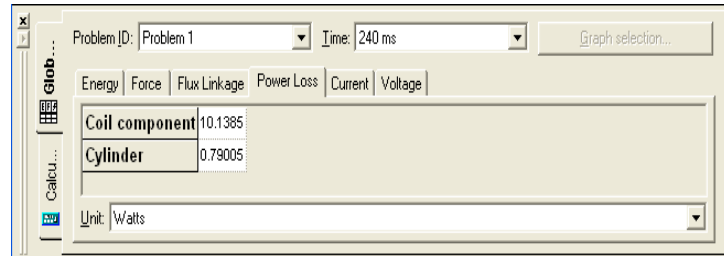
## Save the animation

1. On the File menu, click Save.  
The Save As dialog box appears.
2. In the File Name text box, enter **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.

## View the instantaneous power

The Post Processing bar is automatically displayed at the bottom of the Main window when the solution is complete. The Power page of the Post Processing bar displays the instantaneous power for each conducting component of the model.

1. On the Post Processing bar, click the Global Quantities tab.
2. In the Time drop down list, select 240 (the last time step).
3. Click the Power Loss tab.





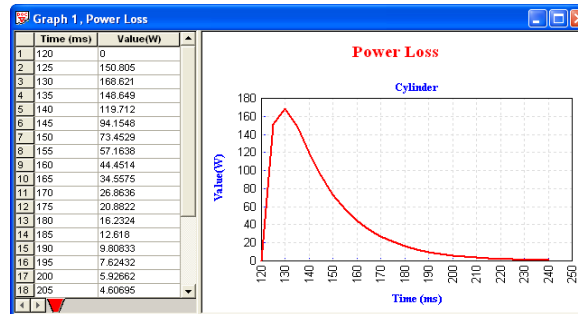
## Graph the power across time

1. Click the mouse pointer in the power entry for the cylinder.

Cylinder	0.79005
----------	---------

2. Press the Graph Selection button.

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



## Save the model

You have now completed the Felix long cylinder.

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

## Summary

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

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



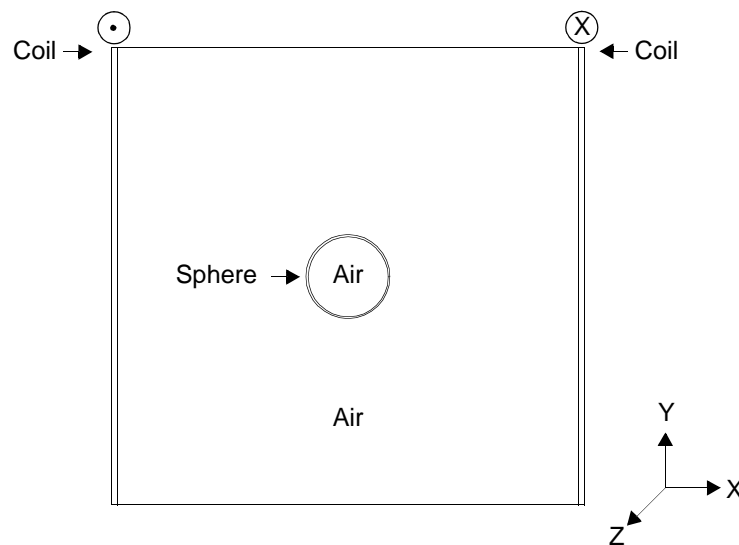
## Chapter 4

# 2D Tutorial: Spherical shield (basic model)

### Modeling plan

The problem is comprised of either a hollow ferromagnetic or a conducting sphere 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 Chapters 2 and 3.

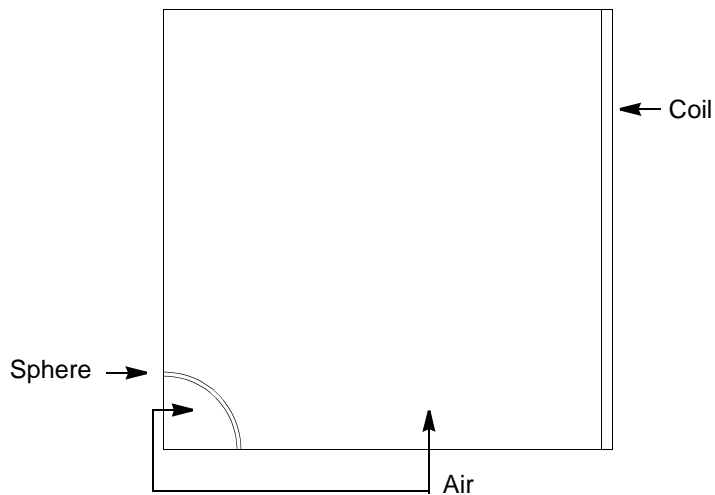


In this tutorial, the basic model is built with the sphere assigned the material AIR. In Chapter 5, the model is solved with a ferromagnetic sphere. In Chapter 6, the model is solved with a conducting sphere.

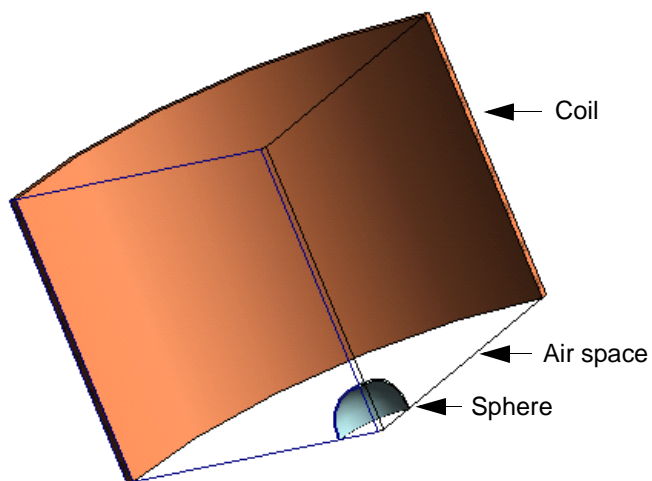
**Tip** In this tutorial, the default rotational sweep of  $90^\circ$  is used to create the one-eighth model. However, you could use any sweep angle to create the model as only the  $Z=0$  plane is solved.

In this tutorial, symmetry conditions are used to model one-eighth of the problem. The model is built from three components: an eighth-sphere, a coil, and an air space.

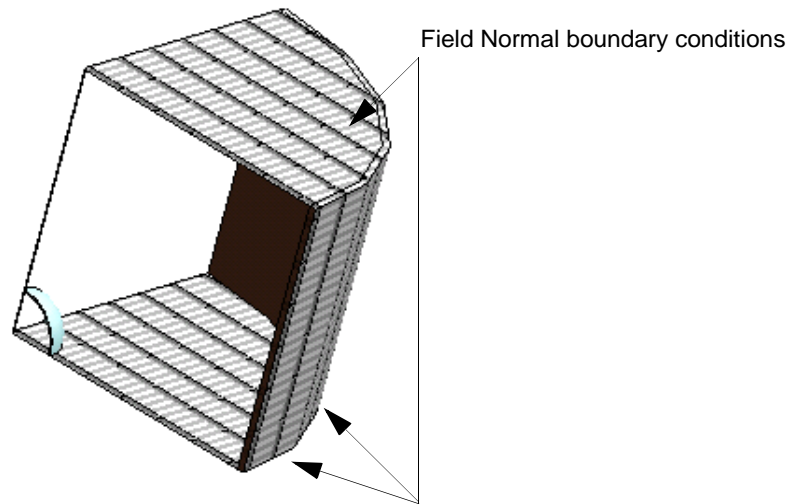
The geometry of the model is shown in the following diagram.



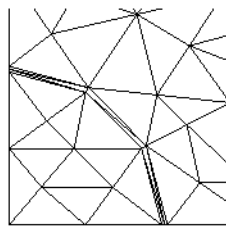
After drawing, the geometry is swept into components and a coil is created from one of the components.



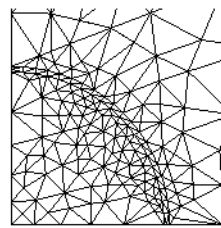
Boundary conditions are applied to indicate symmetry. The Field Normal boundary condition is applied to the top, bottom, and right surfaces of the air space. The default boundary condition, Flux Tangential, is automatically applied to the remaining surfaces.



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

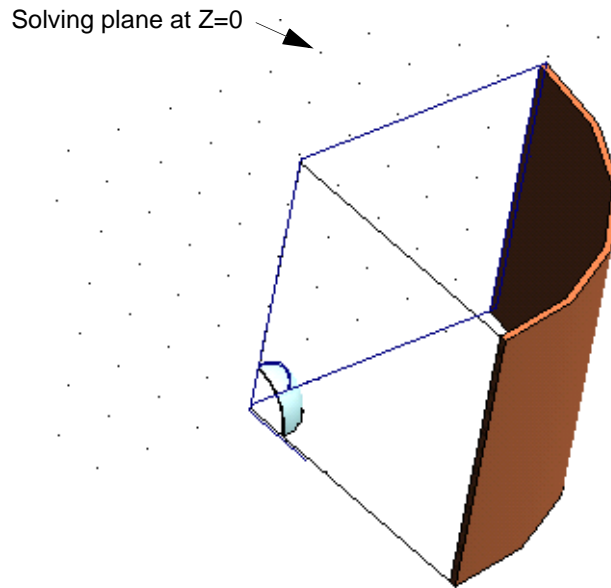


Default initial mesh



Modified initial mesh

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



In this chapter, 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 1 A. Before solving the model, you need to edit the properties of the model as described in the next chapters:

#### **Chapter 5: Magnetostatic**


- Material of the sphere is MU3: Permeability 1000
- Coil has 1000 turns with a current per turn of 63.7 A

#### **Chapter 6: Time-harmonic**

- Material of the sphere is aluminum alloy 6061 with a conductivity of  $2.538e7$  S/m.
- Coil has 1000 turns with a current per turn of 63.7 A
- Source frequency is 400 Hz

## Step 1: Open a new model

---

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

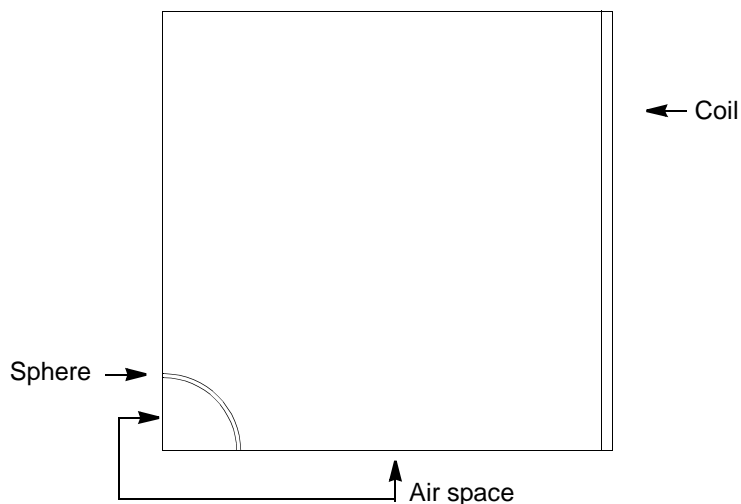
### Name the model

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

## Step 2: Build the geometric model


---

The geometry of the model is shown below.



### Create the air space

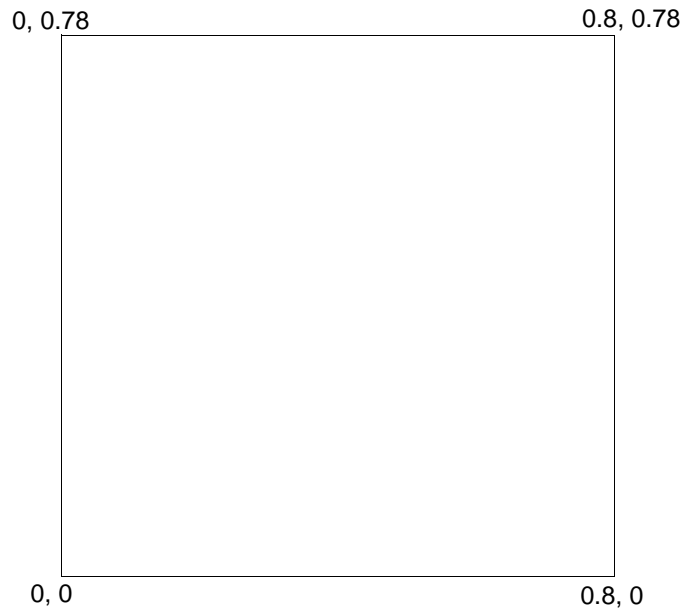
#### Set the drawing area

1. Turn off the display of the Construction Grid (if it is displayed).  
If the grid is visible, select Construction Grid on the View menu.
2. On the Zoom toolbar, click the Update Automatically tool .  
This option updates the display of the model to fit inside the View window.

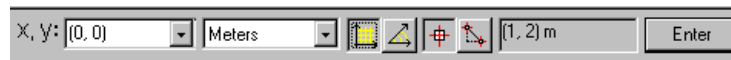





## Draw the geometry of the air space

The geometry of the air space 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 the Line drawing tool .

4. 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


5. Press ESC to stop drawing.

### Make the air space component


**Tip** This sweep angle is equivalent to  $-90^\circ$  around the positive Y axis.

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

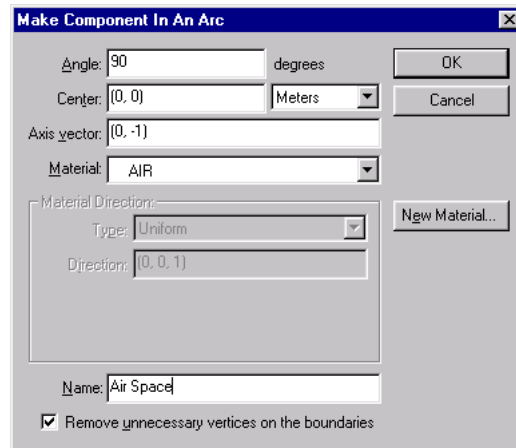
Components are created using the Make Component dialog box.

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

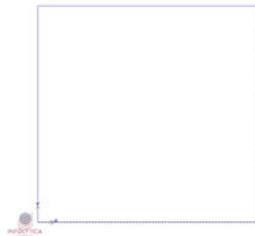
Or, on the Edit menu, click Select Construction Slice Surfaces.
2. Click the mouse pointer inside the surface of the air space. The surface is highlighted when selected.

3. On the Model toolbar, click the Make Component in an Arc tool .

The Make Component In An Arc dialog box appears.



4. In the Make Component In An Arc dialog, enter the following data:  
 Angle: **90**  
 Center: **(0, 0) Meters**  
 Axis vector: **(0, -1)**  
 Material: **Air**  
 Name: **Air Space**
5. Click OK to accept the settings.  
 The component is created.




6. On the File menu, click Save.

## Create the sphere

The quarter-sphere is positioned at the lower left corner of the air space. It has a wall thickness of .0048 meters and an inner radius of 0.1317 meters.



## Draw the sphere


1. On the Model toolbar, click the Arc drawing tool .
2. In the Keyboard Input bar, enter the following coordinates for the inner arc of the sphere.

**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

3. In the Keyboard Input bar, enter the following coordinates for the outer arc of the sphere.


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

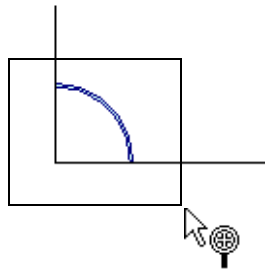
4. On the Draw toolbar, click the Line drawing tool .
5. In the Keyboard Input bar, enter the following coordinates to draw the lines of the sphere.
 

Start coordinates	0, 0.1365	Press ENTER
End coordinates	0, 0.1317	Press ENTER
6. Press ESC.
7. In the Keyboard Input bar, enter the following coordinates to draw the lines of the sphere.
 

Start coordinates	0.1317, 0	Press ENTER
End coordinates	0.1365, 0	Press ENTER
8. Press ESC.

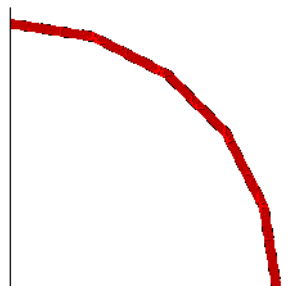
### Make the component of the sphere


1. On the Zoom toolbar, click the Zoom In tool .
2. Drag the mouse pointer 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.

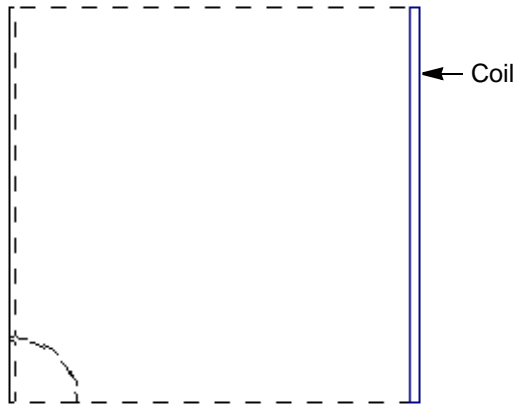
5. Click the mouse pointer inside the surface of the sphere.  
The surface is highlighted when selected.





6. On the Model toolbar, click the Make Component in an Arc tool .
7. 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.
8. In the Name box, enter **Sphere**.
9. Click OK to accept the settings.
10. On the File menu, click Save.

## Create the coil component

The geometry of the coil component is shown below.




### Draw the coil

1. On the Zoom toolbar, click the View All tool .  
The entire model is displayed.
2. On the Draw toolbar, click the Line drawing tool .
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.

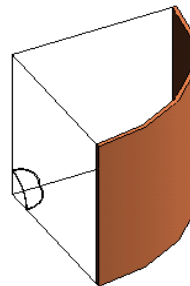
### Make the component of the coil

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 the Make Component in an Arc tool .
4. In the Material drop down list, select Copper: 5.77e7 Siemens/meter.
5. In the Name box, enter **Coil component**.
6. Click OK to accept the settings.
7. On the File menu, click Save.

### Rotate the display of the model

1. On the Preset View toolbar, click the Dynamic Rotation tool .
2. Click the mouse pointer on the 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.

The display is rotated about the center of the model. The model is displayed as a wireframe during the rotation.





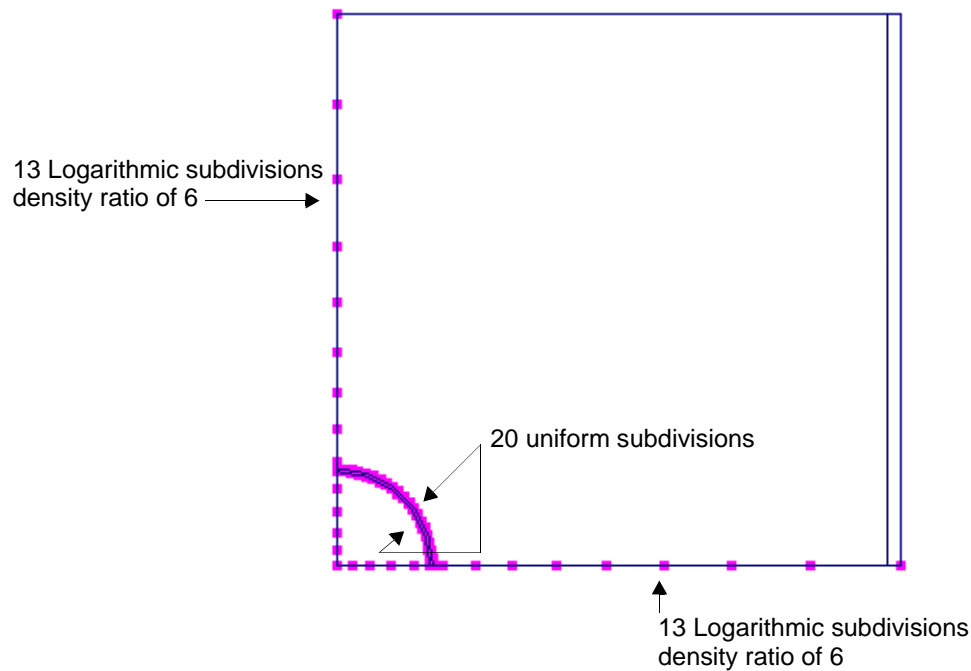
## Step 3: Modify the mesh

---

In the finite element method of analysis, the model is divided into a mesh of triangular-shaped 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 small elements.

One method of increasing mesh density is to subdivide component edges into segments. The number of edge segments corresponds to the number of elements along the edge. Note that the subdivisions propagate throughout the entire model in the sweep direction.

The edge subdivisions that will be modified in this model are shown in the diagram below.




## View the initial mesh

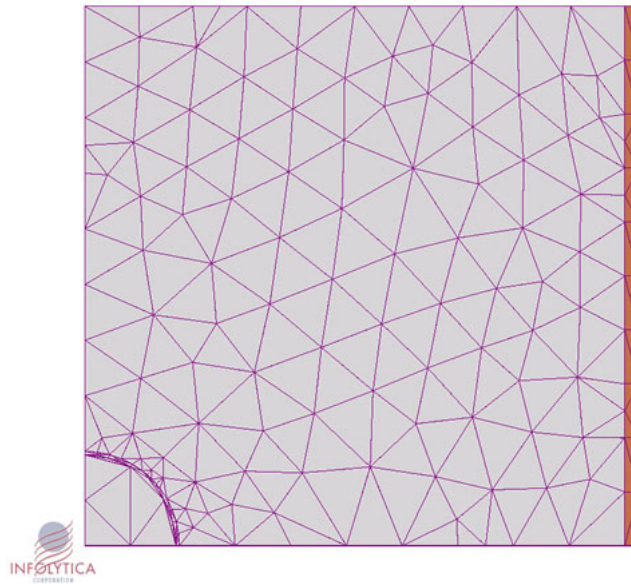
Before changing the subdivisions, the default initial mesh can be viewed.

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

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

2. On the Preset View toolbar, select  (positive Z axis).

The mesh should look like the following diagram.



## Modify the edge subdivisions

Subdivisions can be applied to an edge using either a logarithmic or uniform (linear) scale. Note that subdivisions are edited on the model, not directly on the mesh. Subdivisions are applied to the model using the Mesh toolbar.



1. In the Object page, click the plus sign (+) beside the name of the model (Spherical Shield).



The tree directory expands.




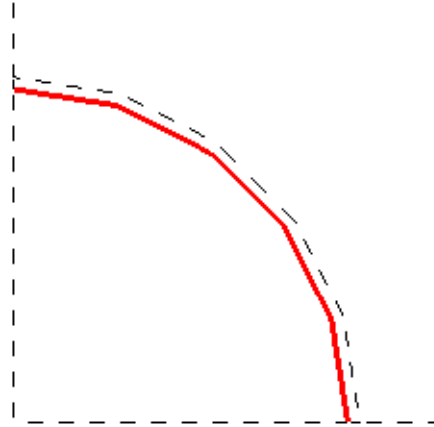
2. In the Object page, click the right mouse button on Coil component.  
A pop-up menu appears.
3. On the pop-up menu, toggle Visible.  
The component is hidden from view. A red "X" appears by the component's name in the Object page.

## Subdivide the edges of the sphere




1. On the View menu, click Solid Model.

**Note** The mesh can be edited only in the Wireframe or Solid Model views.

2. Use the Zoom In tool  to enlarge the area around the sphere.
3. On the Edit menu, click Select Component Edges.
4. Click the mouse pointer on one of the arcs of the sphere.

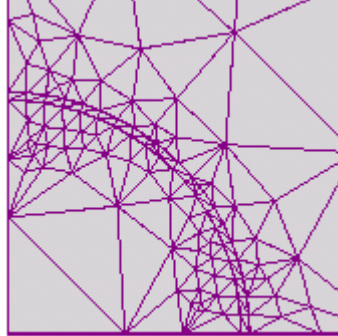


**Tip** When assigning Uniform subdivisions, the Density Ratio is ignored.



5. On the Mesh toolbar, in the Number of Subdivisions text box (the left text box) , enter **20**.
6. On the Mesh toolbar, select the Uniform Subdivision tool . 20 uniform subdivisions are assigned to the edge.
7. Click the mouse pointer on the other arc of the sphere.
8. On the Mesh toolbar, select the Uniform Subdivision tool . 20 uniform subdivisions are assigned to the edge.

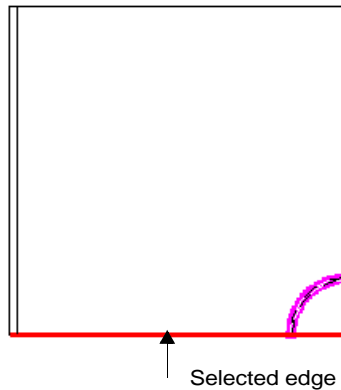
### View the changes to the mesh


- On the View menu, click Initial 2D Mesh.  
The updated mesh should look like the following diagram.



### Subdivide the edges of the air space


- On the View menu, click Wireframe Model.
- On the Preset toolbar, click  (negative Z axis).  
The edges are easier to select when the model is rotated in this way.
- On the Zoom toolbar, click the View All tool .
- Click the mouse pointer on the edge of the air space (as shown in the following diagram).



5. On the Mesh toolbar, in the Number of Subdivisions text box (the left text box) , enter **13**.
6. In the Subdivision Density Factor text box (the right text box), enter **6**. The Mesh toolbar should look like the diagram below.




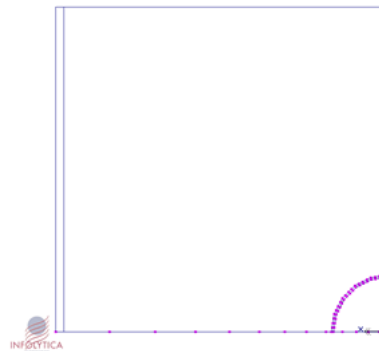
The density factor determines the length of the longest segment in relation to the shortest segment. For example, a density factor of 6 means that the longest segment is 6 times longer than the shortest segment.

7. On the Mesh toolbar, click the Logarithmic: Concentrated at End tool .

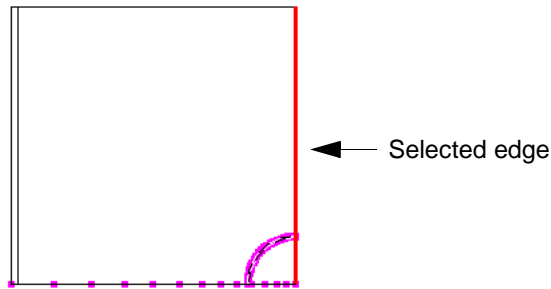
This setting places more subdivisions at the end of the edge than at the start of the edge.


The subdivisions should look like the following diagram.

**Tip** If the subdivisions are concentrated at the other end of the edge, click the Logarithmic: Concentrated at Start tool .




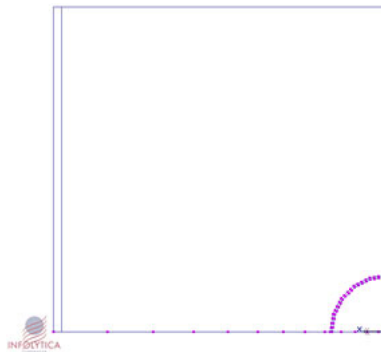
8. Click the mouse pointer on the edge of the air space (as shown in the following diagram).



9. On the Mesh toolbar, click the Logarithmic: Concentrated at Start tool .


This setting places more subdivisions at the start of the edge than at the end of the edge. The subdivisions should look like the following diagram.

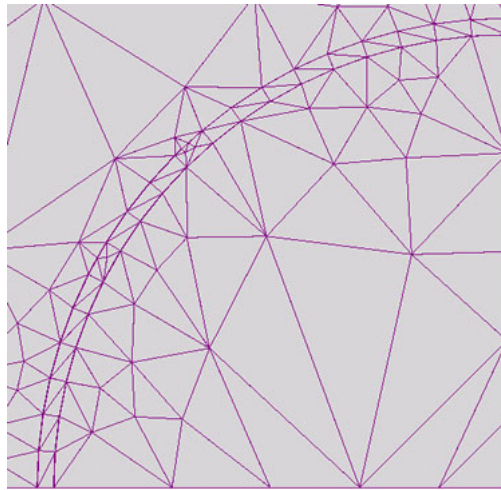
**Tip** If the subdivisions are concentrated at the other end of the edge, click the Logarithmic: Concentrated at End tool .



10. Click Save.

**View the changes to the mesh**

1. On the View menu, click Initial 2D Mesh.  
The mesh updates.
2. Use the Zoom In tool  to enlarge the area around the sphere.  
The mesh should look like the following diagram.







## Step 4: Define boundary conditions

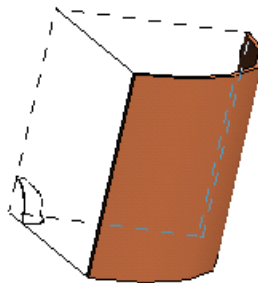
---

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

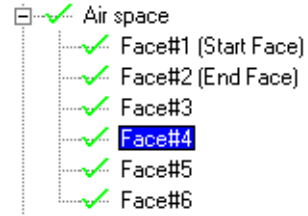
The Field Normal boundary condition constrains to zero the tangential component of the field. The field is made normal (perpendicular) to the boundary. The Flux Tangential boundary condition constrains to zero the normal component of the magnetic flux density. The flux is made to flow tangential to (along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

### Apply the Field Normal boundary condition

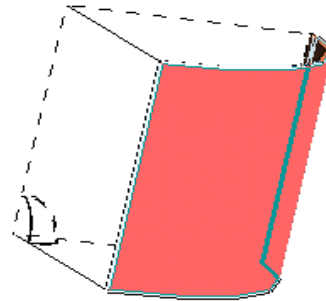
1. On the View menu, click Solid model.
2. On the Object page, toggle the visibility of Coil component so that the component is visible. (Right-click Coil component and toggle Visible on the pop-up menu.)
3. On the Zoom toolbar, click the View All tool .
4. On the Preset View toolbar, click the Dynamic Rotation tool .
5. 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.



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

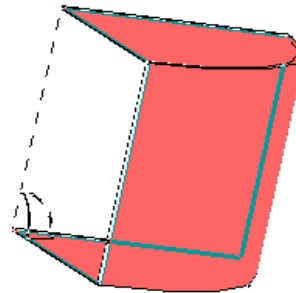



The surface is selected in the View window.

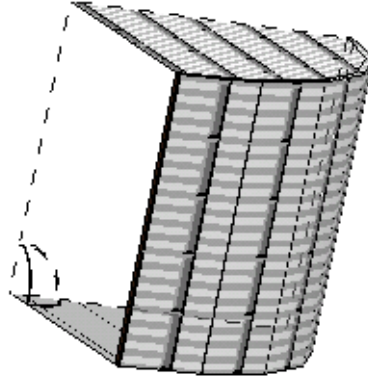


7. While holding down the CTRL key on your computer keyboard, also select Air space, Face#3 and Face#5 .

The three surfaces are selected in the View window.



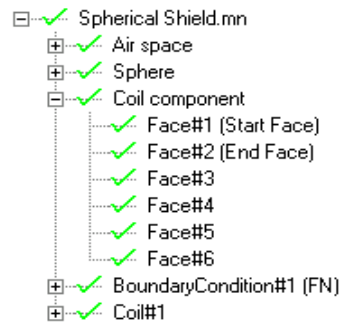
8. On the Boundary Condition toolbar, select Field Normal . The Field Normal boundary condition is applied to the selected surfaces.



## Step 5: Create the coil

---

1. On the Object page, select Coil component, Face#1 (Start Face).
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.

## Summary

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

- Drawing with the Keyboard Input bar
- Creating components
- Rotating the display of a model
- Modifying the default mesh edge subdivisions
- Defining boundary conditions
- Creating a coil

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

Chapter 5: Spherical Shield: Magnetostatic version

Chapter 6: Spherical Shield: Time-harmonic version

## Chapter 5

# 2D Magnetostatic tutorial: Spherical shield

In this tutorial, the spherical shield modeled in Chapter 4 is edited to display the following properties:

- Material of the sphere is MU3: Permeability 1000
- Coil has 1000 turns with a current per turn of 63.7 A

After solving, the value of the magnitude of  $B$  in the center of the sphere is obtained and the contour plot is viewed.

## Step 1: **Copy the basic model**

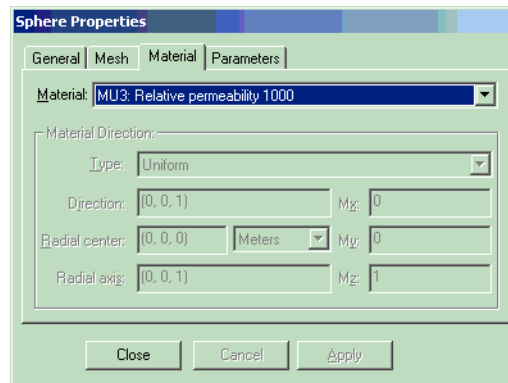
---

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 Spherical Shield model.
3. Select Spherical Shield.
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. Click Save.

## Step 2: Change the material of the sphere

---

1. On the Object page, select the Sphere component.
2. On the Edit menu, click Properties.  
The Component property dialog is displayed.
3. Select the Material tab.  
The dialog shows that the material currently applied to the component is AIR.



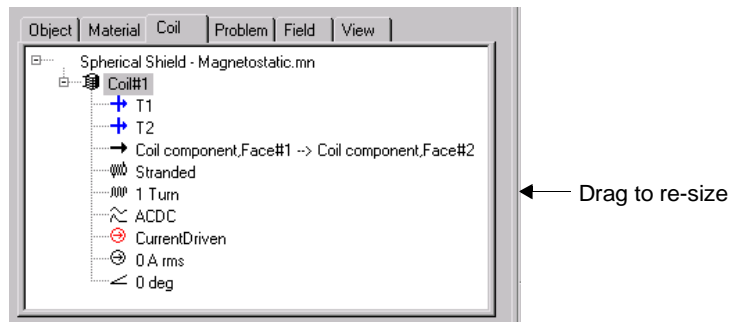
4. In the Material drop down list, select MU3: Relative permeability 1000.
5. Click OK.  
The material is applied to the component.

## Step 3: Edit the coil properties

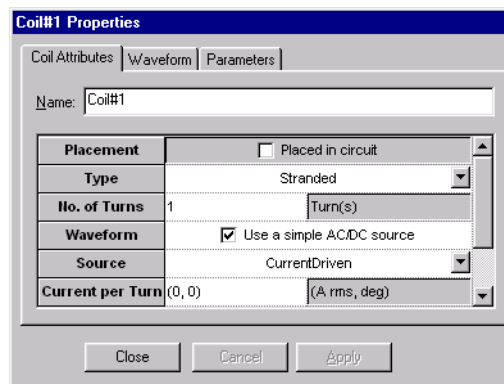
1. On the Project bar, click the Coil tab.

The Coil page is displayed.

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



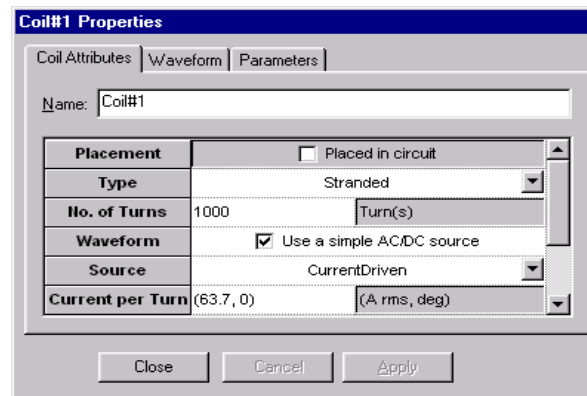
2. Select the name of the coil (Coil#1).
  3. On the Edit menu, click Properties.
- The Coil property dialog appears.



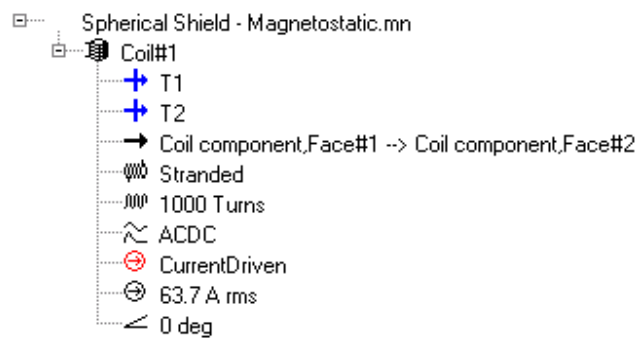
4. In the No. of Turns box, enter **1000**.
5. Press ENTER on your keyboard.



6. In the Current Per Turn box, enter **(63.7, 0)**.  
The current is 63.7, the phase is 0.



7. Click OK.  
The Coil page is automatically updated.

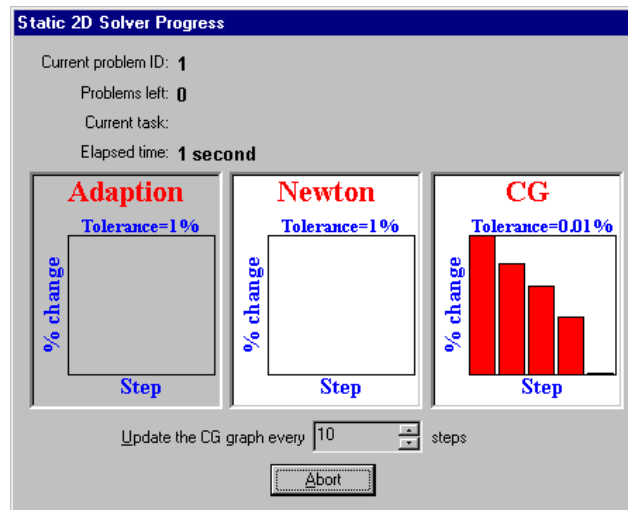


8. Click Save.

## Step 4: Solve

---

- On the Solve menu, click Static 2D.  
The Static 2D Solver Progress dialog appears.



The spherical shield takes less than 10 seconds to solve (solving time may vary according to computer). The Solver Progress dialog automatically exits when the solution is complete.

## Step 5: View the solution results

---


The Post Processing bar is automatically displayed at the bottom of the Main window when the solution is complete.

The following results will be reviewed in this section:

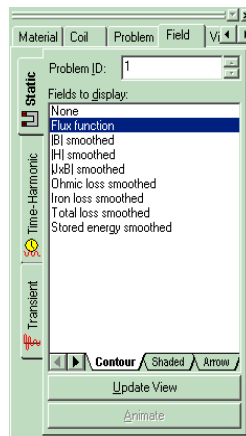
- The contour plot
- The magnitude of B at (0, 0)

### View the contour plot

The contour plot displays contour lines of the magnetic flux function. These contour lines 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. On the Project bar, select the Field tab. (Use the arrow on the Project bar  to scroll the pages if necessary.)

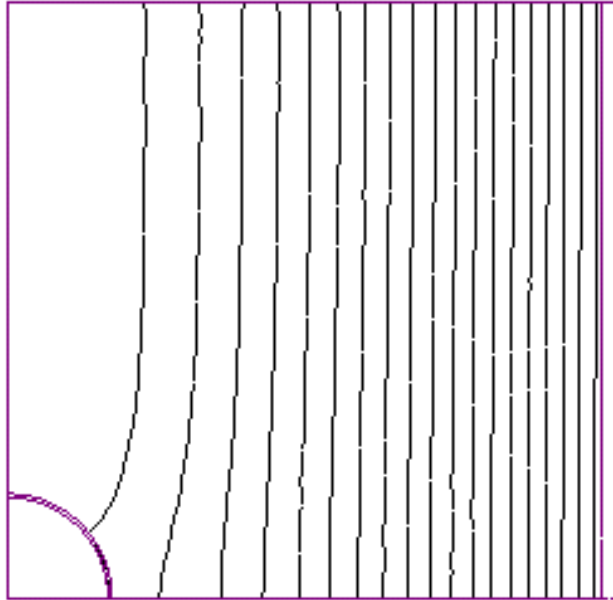
The Field page opens.



2. Click the Contour Plot tab located near the bottom of the Field page.

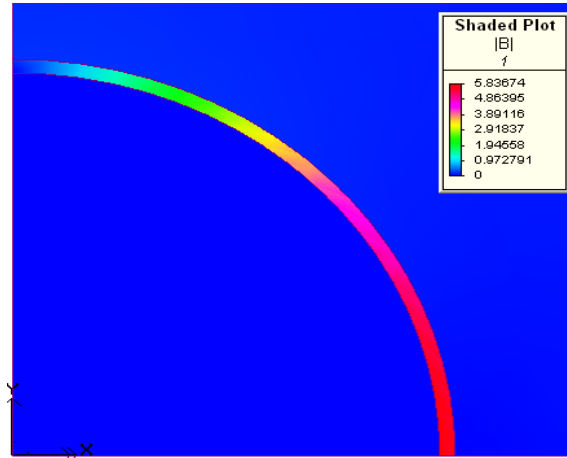
3. In the Fields To Display list, select Flux Function.
4. At the bottom of the Field page, press Update View.  
The contour plot is displayed. (This may take a moment.)  
The contour plot should look like the following diagram.

**Tip** If the View window is not displaying the entire contour plot, click the View All tool  on the Zoom toolbar.



## View the shaded plot of $|B|$

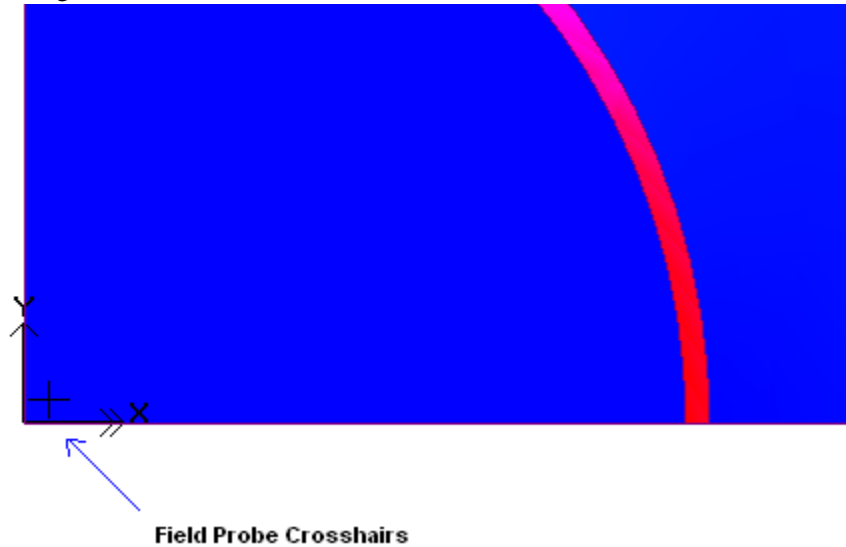
1. On the Project bar, select the Field tab.
2. Select the Contour page.
3. In the Fields To Display list, make sure that **None** is selected.
4. Select the Shaded page.
5. In the Fields To Display list, select  $|B|$ .
6. Select Update View.



The shaded plot on the surface of the model is displayed. (This may take a moment.) A color legend is displayed beside the shaded plot

## Probe the magnitude of 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 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.



### To probe for field values using the mouse

1. On the Tools menu, click Field Probe.  
A checkmark indicates that it is enabled.
2. Move the mouse (crosshairs) over the solution near (0,0).  
The field value, for that specified location on the solution, is displayed in the Status Bar.

B = 0.00576085	m	X: N/A	Y: N/A	Xg:0.000703	Yg:0.001718	Zg:0
----------------	---	--------	--------	-------------	-------------	------

← |B| value at these coordinates  
as shown on the Status bar →

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

The Text Output Bar automatically opens (if it isn't already), and the field value and its location (x, y, and z coordinates) on the solution is displayed.

X	Y	Z	B
0.000978722	0.000752079	0	0.00576086
0.00322941	0.0026122	0	0.00576087
0.00585522	0.00335625	0	0.00576088
0.00623033	0.00372827	0	0.00576089
0.00360453	0.000380055	0	0.00576087
0.000228492	0.00186815	0	0.00576085
0.000228492	8.03142e-006	0	0.00576085
0.000603607	8.03142e-006	0	0.00576085

The example above shows several field values (under the heading |B|) and their location for every area that was clicked upon.

## Save the model

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

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

## 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 the Contour plot of the model
- Viewing the Shaded plot of the model
- Probing a field value using the Field Probe feature

### Further exploration

In this tutorial, solution accuracy was improved by increasing the mesh density around the sphere. Other possible options include:

- Using 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.



## Chapter 6

# 2D Time-harmonic Tutorial: Spherical shield

In this tutorial, the spherical shield modeled in Chapter 5 is edited to display the following properties:

- Material of the sphere is Aluminum 6061 with a conductivity of  $2.538e7$  S/m.
- Coil has 1000 turns with a current per turn of 63.7 A
- Source frequency is 400 Hz

After solving, the power and the contour plot are viewed. The magnitude of  $B$  is probed at the center of the sphere (0,0).

## Step 1: **Copy the basic model**

---

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 Spherical Shield model.
3. Select **Spherical Shield - Magnetostatic**.
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. Click Save.

## Step 2: Create a new material

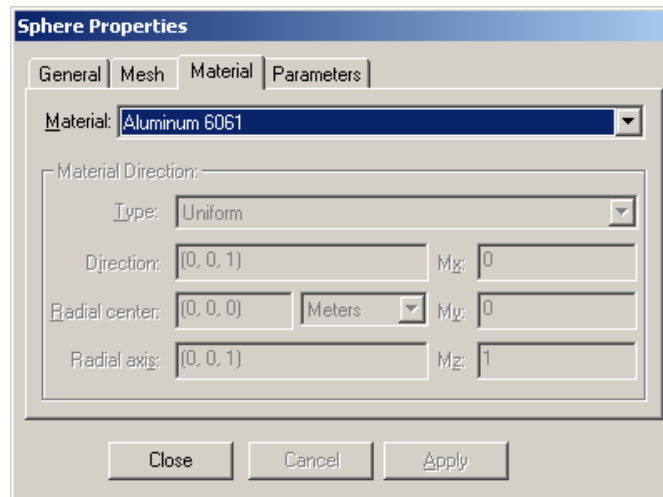
---

**Note** If you have already created Aluminum 6061 in Chapter 2 (Cylindrical Shield - Time-Harmonic tutorial), then you can skip this step and proceed to Step 3 on page 108.

1. On the Tools menu, click New User Material.
2. On the General page, enter the following data:
  - Name: **Aluminum 6061**
  - Display color: *Click Set Color and select an appropriate color*
  - Description: *Optional*
3. Click Next.
4. On the Options page, select the following:
  - Magnetic ***Permeability***
  - Electric ***Conductivity***
5. 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**
6. Once you have entered all the values, click Finish in the Confirmation page to create the new material.
7. On the File menu, click Save.

## Step 3: **Change the material of the sphere**

1. On the Object page, select the Sphere component.
2. On the Edit menu, click Properties.  
The Component property dialog is displayed.
3. Select the Material tab.

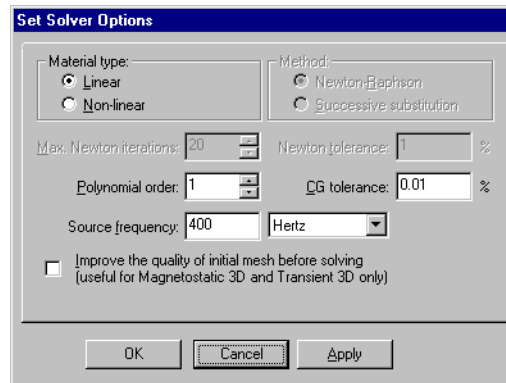


4. In the Material drop down list, select **Aluminum 6061**.
5. Click OK.  
The material is applied to the component.

## Step 4: Set the source frequency

---

1. On the Solve menu, click Set Solver Options.  
The Set Solver Options dialog appears.



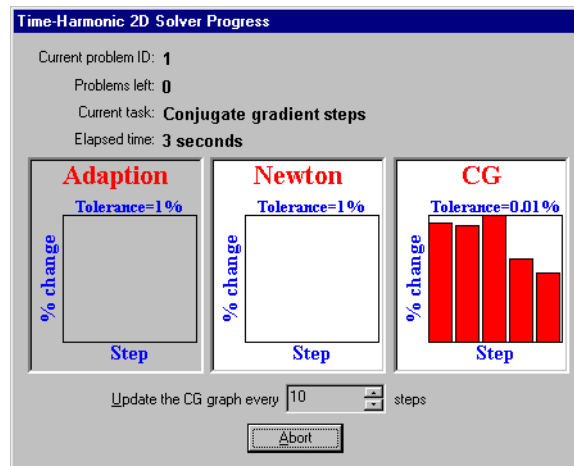
**Tip** If the Linear option is not set, the solver will automatically detect the presence of linear materials.

2. Select the Linear option.
3. In the Source Frequency box, enter **400** Hz.  
The default unit is Hertz.
4. Click OK.

## Step 5: Solve

---

- On the Solve menu, click Time-harmonic 2D.  
The Time-harmonic 2D Solver Progress dialog appears.



The spherical shield takes less than 10 seconds to solve (solving time may vary according to computer). The Solver Progress dialog automatically exits when the solution is complete.


## Step 6: View the solution results

The following results will be reviewed in this section:

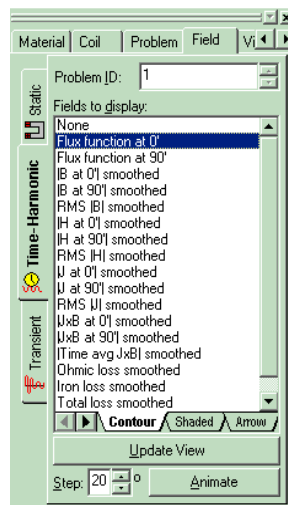
- The contour plot
- The time-averaged Ohmic power dissipated in the conductor
- The magnitude of  $B$  at  $(0, 0)$

### View the contour plot

The contour plot displays contour lines of the magnetic flux function. These contour lines are the magnetic flux lines (lines that are everywhere parallel to the flux density vector).


1. On the Project bar, select the Field tab. (Use the arrow on the Project bar  to scroll the pages if necessary.)

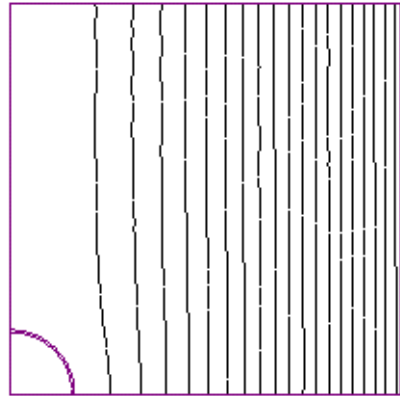
The Field page opens.



2. Click the Contour plot tab (at the bottom of the Field page).
3. In the Fields To Display list, select Flux Function at  $0^\circ$ .

4. At the bottom of the Field page, press Update View.  
The contour plot is displayed. (This may take a moment.)

**Tip** If the View window is not displaying the entire contour plot, click the View All tool  on the Zoom toolbar.



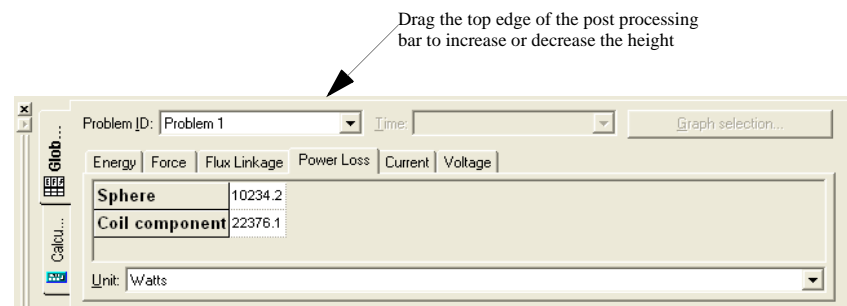
## View the time-averaged Ohmic power

The Post Processing bar is automatically displayed at the bottom of the Main window when the solution is complete.

1. On the Post Processing bar, click the Global Quantities tab.
2. Click the Power Loss tab.

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

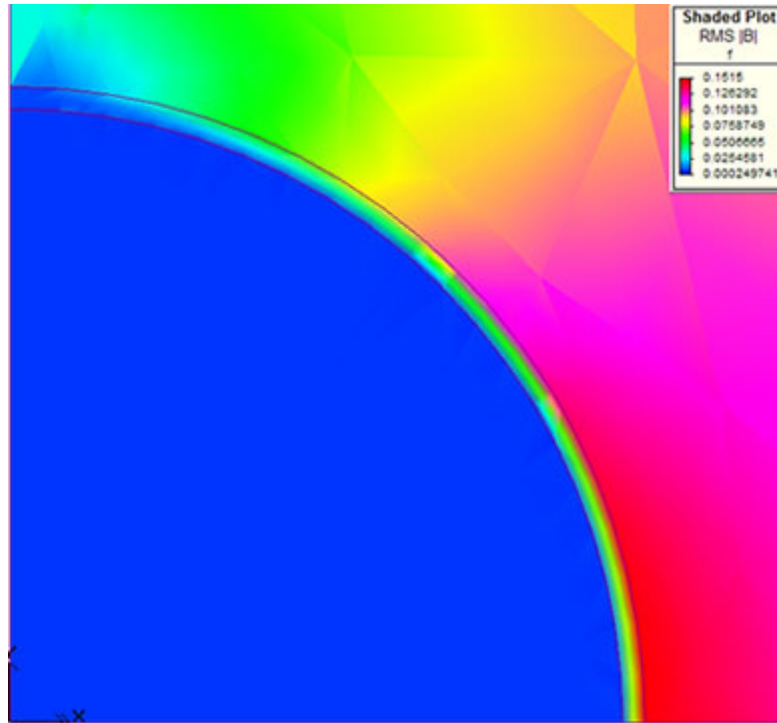
**Tip** Adjust the height of the Post Processing bar by placing the cursor on its top edge, then while keeping the left mouse button pressed, dragging the bar up or down.





## View the shaded plot of $|B|$

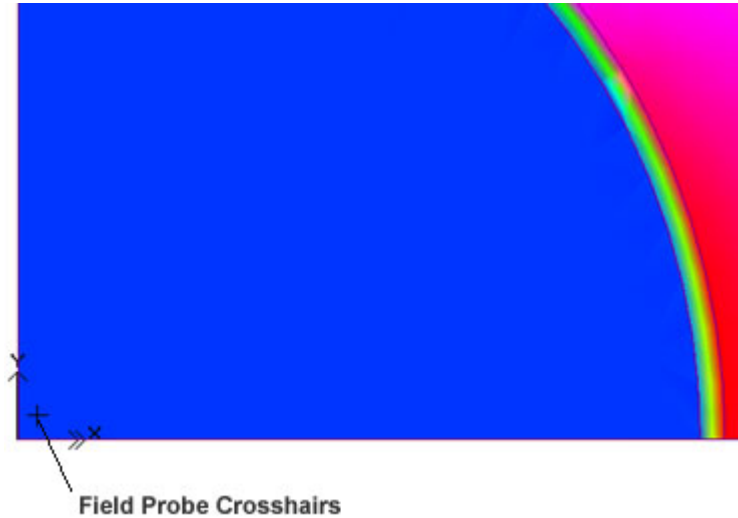
1. On the Project bar, select the Field tab.
2. Select the Contour page.
3. In the Fields To Display list, make sure that **None** is selected.
4. Select the Shaded page.
5. In the Fields To Display list, select **RMS  $|B|$** .
6. Select Update View.



The shaded plot on the surface of the model is displayed. (This may take a moment.) A color legend is displayed beside the shaded plot

## Probe the magnitude of 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 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.



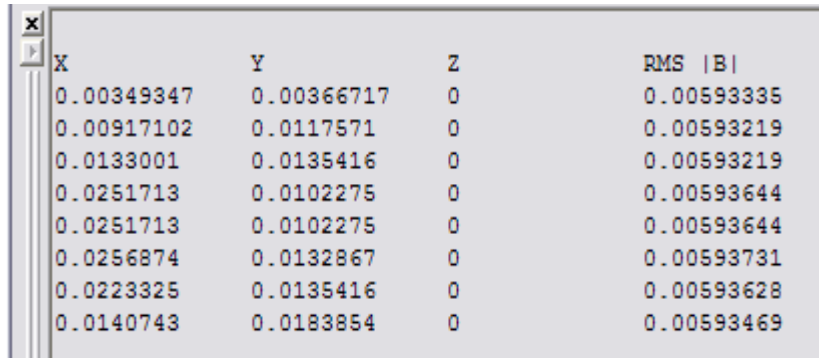
### To probe for field values using the mouse

1. On the Tools menu, click Field Probe.  
A checkmark indicates that it is enabled.
2. Move the mouse (crosshairs) over the solution near (0,0).  
The field value, for that specified location on the solution, is displayed in the Status Bar.

RMS |B| = 0.00593335      m    X: N/A    Y: N/A    Xg: 0.000158   Yg: 0.000903   Zg:

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

The Text Output Bar automatically opens (if it isn't already), and the field value and its location (x, y, and z coordinates) on the solution is displayed.



X	Y	Z	RMS  B
0.00349347	0.00366717	0	0.00593335
0.00917102	0.0117571	0	0.00593219
0.0133001	0.0135416	0	0.00593219
0.0251713	0.0102275	0	0.00593644
0.0251713	0.0102275	0	0.00593644
0.0256874	0.0132867	0	0.00593731
0.0223325	0.0135416	0	0.00593628
0.0140743	0.0183854	0	0.00593469

The example above shows several field values (under the heading |B|) and their location for every area that was clicked upon.

## Save the model

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

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

## 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
- Editing the properties of a coil
- Setting the source frequency
- Setting the linear solving option
- Viewing the Contour plot of the model
- Viewing the time-averaged Ohmic power loss in the conductor
- Viewing the Shaded plot of the model
- Probing a field value using the Field Probe feature

### Further exploration

In this tutorial, solution accuracy was improved by increasing the mesh density around the cylinder. 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.

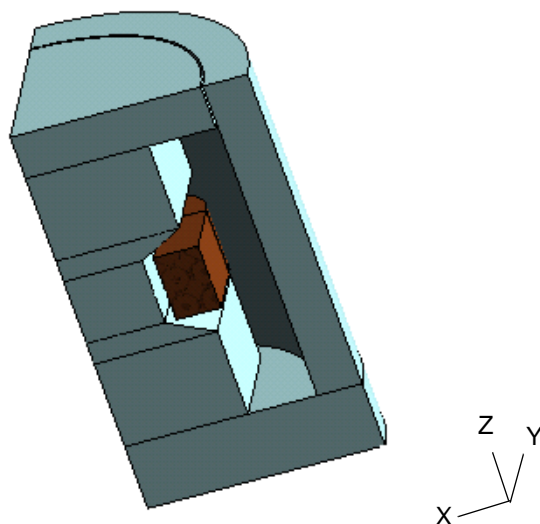
## Chapter 7

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

### 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.)



**Note** It is recommended that you complete the 3D “Quick Start” Tutorial in the *Getting Started Guide* before beginning the pot-core tutorial.

## Step 1: Open a new model

---

- Start MagNet.

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

### 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.

### Set the model units

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

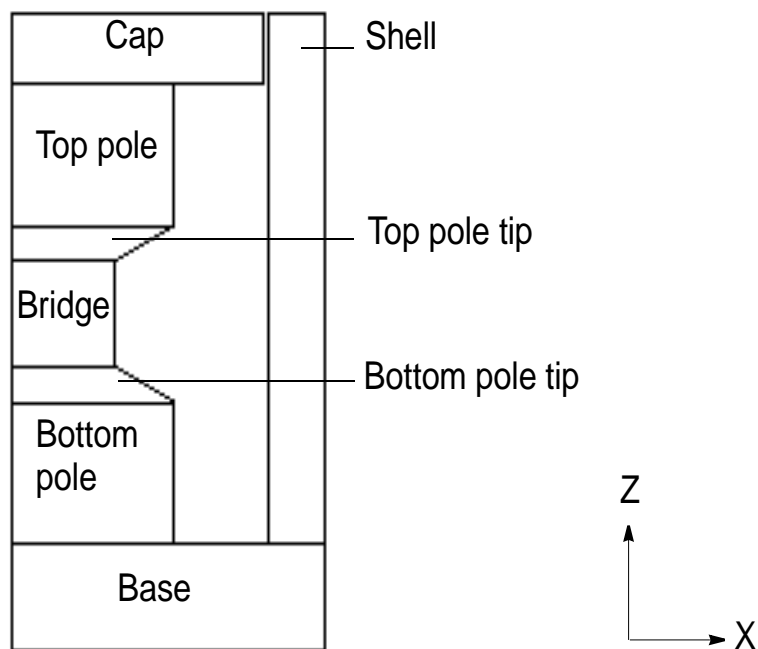
You can set centimeters to be preferred unit of measurement in all the MagNet dialogs. This option is set in the General Model property page.

1. In the Object page of the Project bar, select the name of the model (in this case, 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.

## Step 2: Build the geometric model

---

The pot-core is built from eight components.



## Create the base of the pot-core

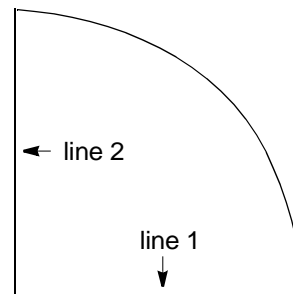
The first component forms the base of the pot-core.




### 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 the Line drawing tool .
4. 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

5. Press ESC.




6. 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

7. Press ESC.

8. On the Draw toolbar, click the Arc drawing tool .



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

Center coordinates      0, 0      Press ENTER

Start coordinates      10.6, 0      Press ENTER

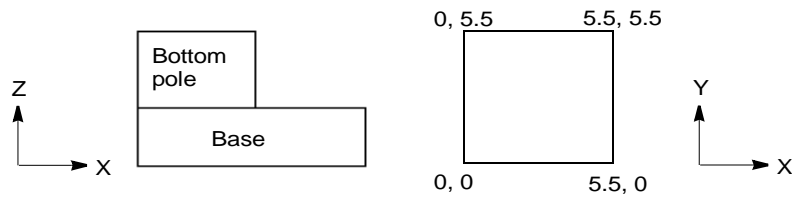
End coordinates      0, 10.6      Press ENTER

### Make the component

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


## 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.

### Move the construction slice

1. On the Object page, select Base, Face#2 (End Face).
2. On the Draw toolbar, click the 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.

### Draw the geometry

1. On the Draw toolbar, click .
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.

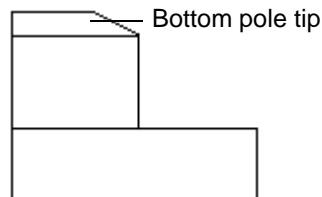
## Make the component

The instructions for making the component are now abbreviated. If you need help with the steps, please refer to page 121.

1. Select the construction slice surface of the bottom pole.
2. Sweep the component **4 centimeters**.
3. Apply **MU3: Relative permeability 1000**.
4. Name the component **Bottom pole**.
5. Click OK.


## 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.



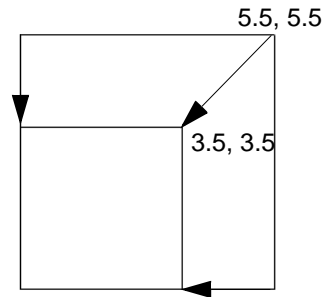
The pole tip is created in two steps. In the first step, a component is created from the surface of the bottom pole. In the second step, the geometry of the new component is distorted.

## Make the component

1. On the Object page, select Bottom pole, Face#2 (End Face).
2. On the Model toolbar, click .
3. Sweep the component **1 centimeter**.
4. Apply **MU3: Relative permeability 1000**.
5. Name the component **Bottom pole tip**.
6. Click OK.

### 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.



#### Move the construction slice

- Move the construction slice to the end face of the Bottom pole tip.

#### 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.

#### Distort vertex (0, 5.5)

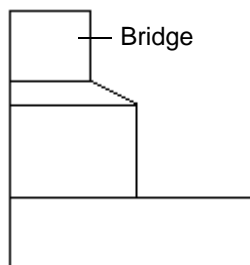
1. In the Keyboard Input bar, type **0, 5.5** (the original coordinate of the selected vertex) and press ENTER.
2. In the Keyboard Input bar, type **0, 3.5** (the new coordinate for the selected vertex) and press ENTER.

**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. Click **Save**.

**Create the bridge between the pole tips**

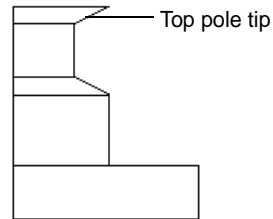
A square block bridges the gap between the pole tips.

**Make the component**

1. Make a component from the end surface (Face#2) of the Bottom pole tip.
2. Sweep the component **3 centimeters**.
3. Apply **MU3: Relative permeability 1000**.
4. Name the component **Bridge**.
5. Click OK.

## Create the top pole tip

The next component is the second pole tip.



### Make the component

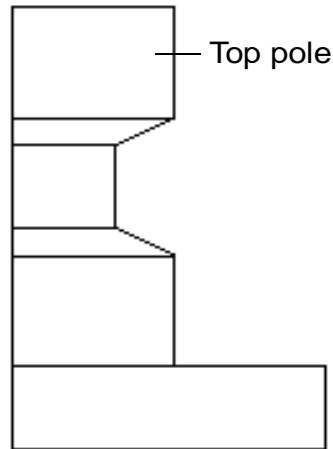
1. Make a component from the end surface (Face#2) of the Bridge.
2. Sweep the component **1 centimeter**.
3. Apply **MU3: Relative permeability 1000**.
4. Name the component **Top pole tip**.
5. Click OK.

### Distort the shape of the top pole tip

1. Move the construction slice to the end surface of the Top pole tip.
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.

## Create the top pole

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



### Make the component

1. Make a component from the end surface of the Top pole tip.
2. Sweep the component **4 centimeters**.
3. Apply **MU3: Relative permeability 1000**.
4. Name the component **Top pole**.
5. Click OK.

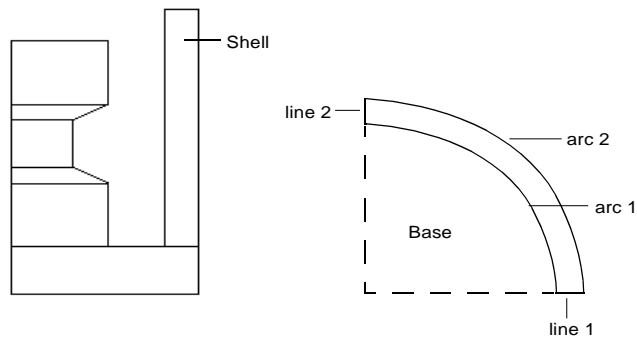
### Delete Construction Slice edges

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.

## Create the shell

The shell begins at the ending surface of the base.



## Draw the geometry

1. Move the construction slice to the end face of the Base.
2. On the Draw menu, click Line.
3. 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



4. On the Draw menu, click Arc.
5. 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

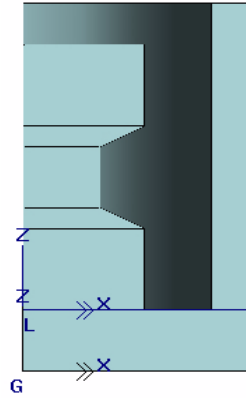
**Make the component**

1. On the Selection toolbar, click .
2. Make a component from the construction slice surface of the shell (the area between the two arcs).
3. Sweep the component **15 centimeters**.
4. Apply **MU3: Relative permeability 1000**.
5. Name the component **Shell**.
6. Click OK.

## Rotate the model

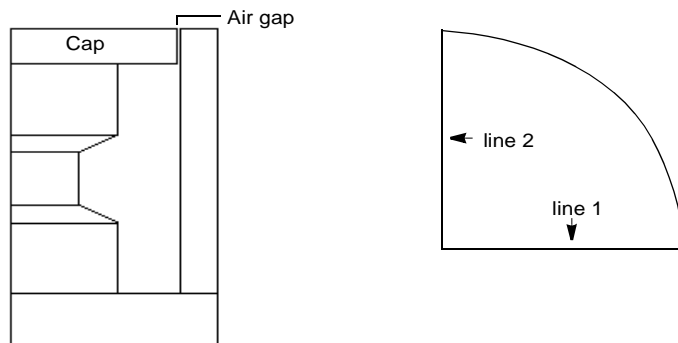
- On the Preset View toolbar, click  to view the model along the negative Y axis.

The model should look like the diagram below.




## 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.



**Draw the geometry**

1. Move the construction slice to the end surface of Shell.
2. On the Preset View toolbar, click  (positive Z axis).
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 ESC

**Line 2:**

Start coordinates      0, 0

End coordinates      0, 8.5

Press ESC

5. On the Draw menu, click Arc.
6. Draw the geometry using the following coordinates.

**Arc:**

Center coordinates      0, 0


Start coordinates      8.5, 0

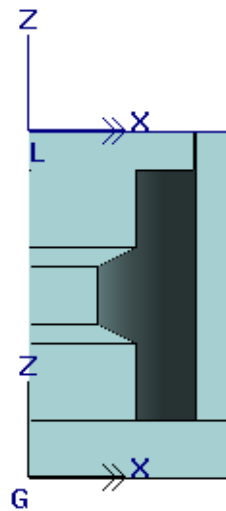
End coordinates      0, 8.5

### Make the component

1. Make a component from the construction slice surface of the cap.
2. Sweep the component **-2 centimeters**.  
Entering a negative number sweeps the component backward (in this case, in the negative Z direction).
3. Apply **MU3: Relative permeability 1000**.
4. Name the component **Cap**.
5. Click Ok.

### Rotate the model

- On the Preset View toolbar, click  (negative Y axis).  
The model should look like the diagram below.



## Step 3: Create the excitation

---

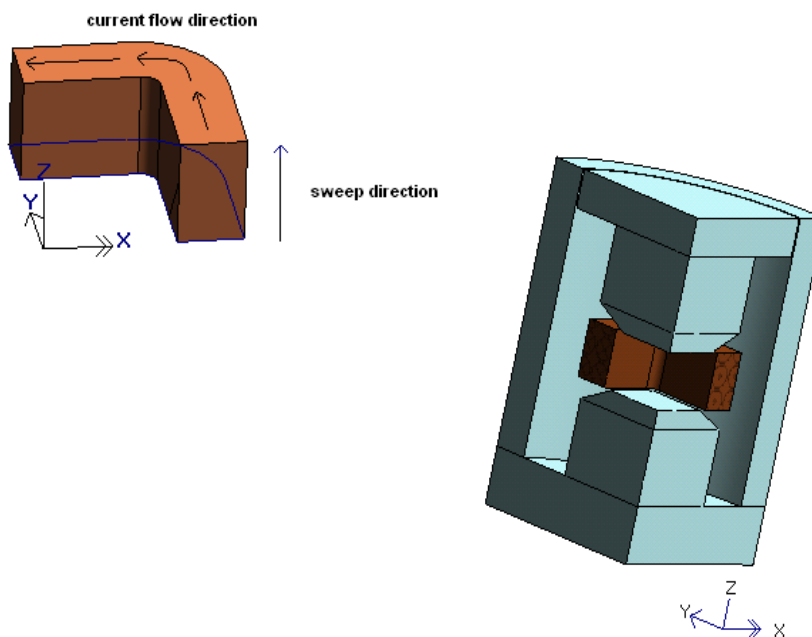
In 3D, simple coils are created by selecting either a component or a component surface and then selecting the Make Simple Coil command. The method chosen to make the coil depends on how the coil has been swept and how you want the current to flow in the coil.

When a simple coil is formed from a connected group of components, each component of the group must share a start or end surface with another component. 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.

Sweep and current flow direction



## Create the component of the coil

### 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.

### 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.

### Display a wireframe view

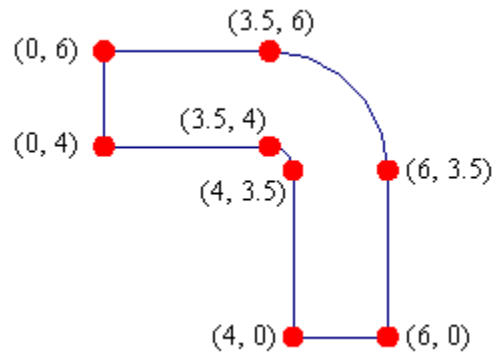
- On the View menu, click Wireframe model.  
In a wireframe display, all the edges of a model are displayed without removing hidden surfaces.

### Rotate the model

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

## 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

Press ESC

**Line 2:**

Start coordinates      3.5, 4

End coordinates      0, 4

End coordinates      0, 6

End coordinates      3.5, 6

Press ESC

4. On the Draw menu, click Arc.
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

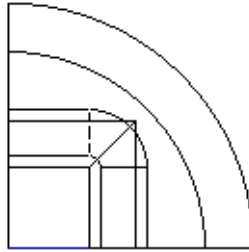



## Make the component

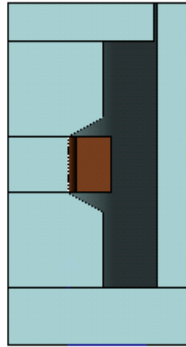
1. On the Edit menu, click Select Construction Slice Surfaces.
2. Click inside the geometric surface you have just created.
3. On the Model menu, click Make Component in a Line.
4. Sweep the component **3 centimeters**.
5. In the Material drop down list, select **Copper: 5.77e7 Siemens/meter**.
6. Name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

## View the coil components

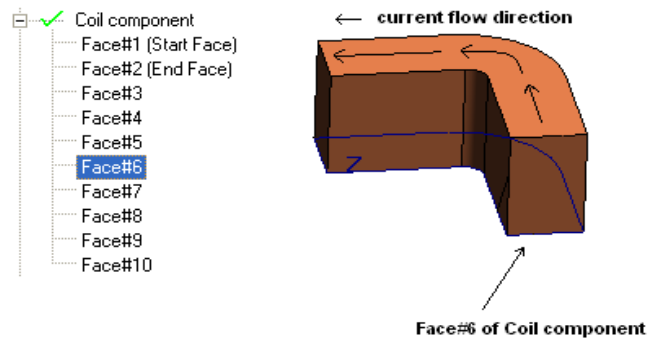
1. On the Preset View toolbar, click  (positive Z axis).



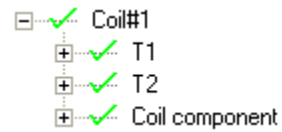
2. After verifying the shape of the coil components, re-display the Shell.
3. Re-display the Solid Model.
4. On the Preset View toolbar, click  (negative Y axis).



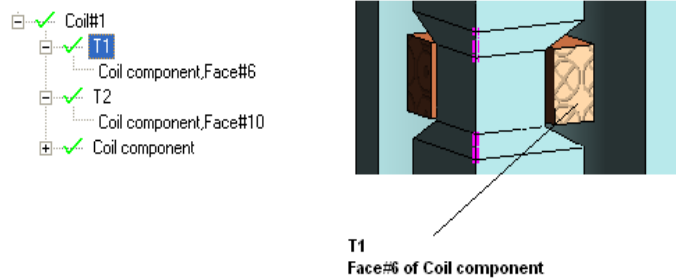
## Create the coil



1. On the Object page, select Face#6 of the Coil component.
2. On the Model menu, click Make Simple Coil.  
The coil is made from all the connecting components. 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.



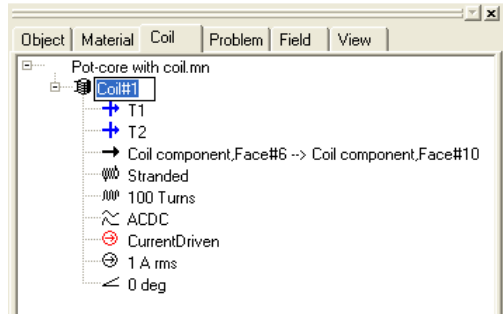
## Change the current of the coil

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

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

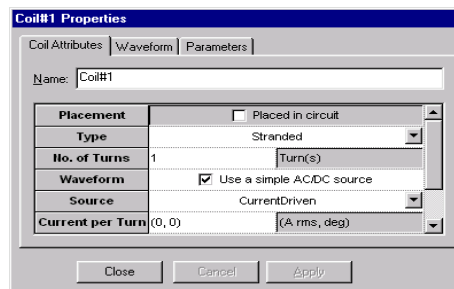
The Coil page appears. Notice that the default number of turns is

1. The default current per turn is 0 A.



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

The Coil Attributes property page appears.



4. In the Number Of Turns text box, enter **100**.
5. In the Current Per Turn text box, enter **(1,0)**.  
The phase is 0.
6. Press OK on the Coil Attributes property page.  
If the OK button is not activated, press ENTER on your computer keyboard.

The properties of the coil are updated on the Coil page.

7. Click Save.

## Step 4: 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.

### 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.

### Create the air box

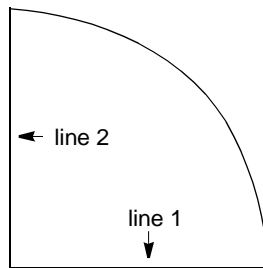
The starting surface of the air box begins 4 centimeters below the starting surface of the base.


### Move the construction slice

1. Move the construction slice to the start surface (Face#1) of Base.
2. Select the Move Construction Slice tool  again.
3. Select Along A Line.
4. In the Distance box, enter -4.
5. Click OK.

## Draw the geometry

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



1. On the Preset View toolbar, select  (positive Z axis).
2. Draw the geometry using the following coordinates.

### Line 1:

Start coordinates	0, 0
End coordinates	13.6, 0
	Press ESC

### Line 2:

Start coordinates	0, 0
End coordinates	0, 13.6
	Press ESC

### Arc:


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

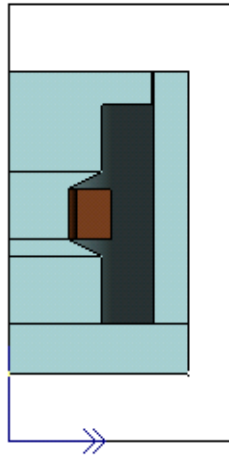
### Make the component

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

1. Make a component from the surface of the air box.
2. Sweep the component **26 centimeters**.
3. Apply the material **AIR**.
4. In the Name box, enter **Air box**.
5. Click Save.

### Rotate the model

- On the Preset View toolbar, click  (negative Y axis).



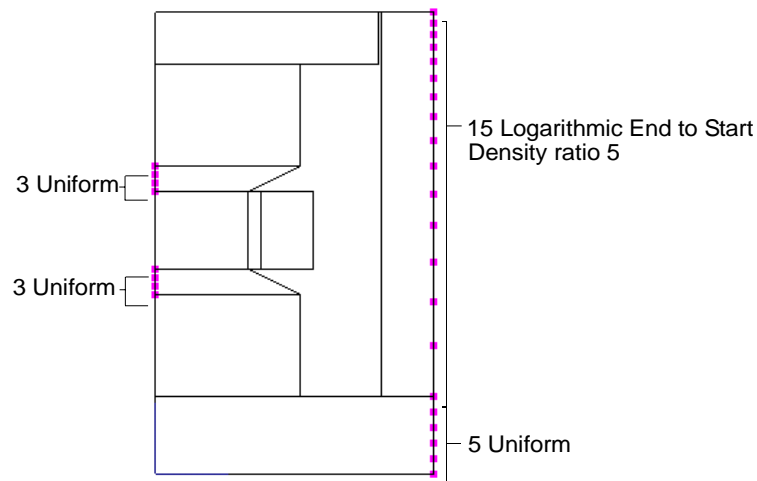
## Step 5: Refine the mesh

---

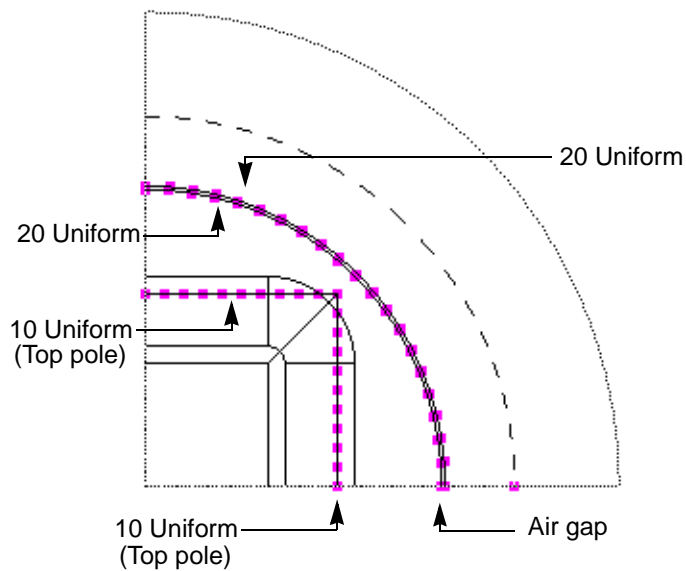
In the finite element method of analysis, the model is divided into a mesh of tetrahedral-shaped elements. The vector field inside each element is represented by a polynomial with unknown coefficients. The finite element analysis is the solution of the set of equations for the unknown coefficients.

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 subdivide component edges into segments. The number of edge segments corresponds to the number of elements along the edge.

The edge subdivisions that will be modified in this model are shown in the diagram below and on page 144.



*Mesh edge subdivisions (negative Y view)*



*Mesh subdivisions (positive Z view)*

## Set the meshing method to extrusion

**Note:** If you do not have the Full 3D Solid Modeler license, this option is not available to you. Your default setting is extrusion.

1. From the Object page, select **Pot-core with coil**.
2. On the Edit menu, click Properties.
3. Select the Mesh page.
4. In the Meshing Method To Use drop down list, select Extrusion.
5. Click Ok.



## Modify the edge subdivisions


Subdivisions can be applied to an edge using either a logarithmic or uniform (linear) scale. Note that subdivisions are edited on the model, not directly on the mesh. Subdivisions are applied to the model using the Mesh toolbar.

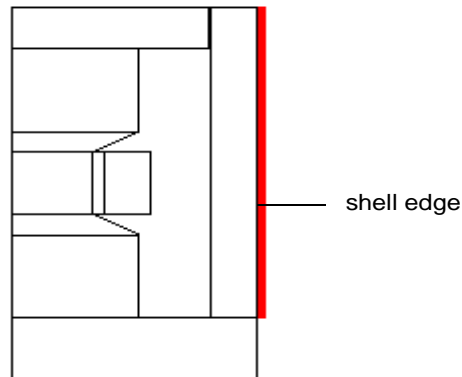



Note also that subdivisions propagate throughout the entire model in the sweep direction.

### Use the logarithmic distribution method

1. On the View menu, click Wireframe Model.
2. On the Edit menu, click Select Component Edges.
3. In the View window, click the mouse pointer on the edge of the shell (as shown in the diagram below).


Use the Zoom In tool  to enlarge the display of the model.



4. On the Mesh toolbar, in the Number of Subdivisions text box (the leftmost text box) , enter **15**.

5. In the Subdivision Density Factor text box (the rightmost text box), enter **5**.

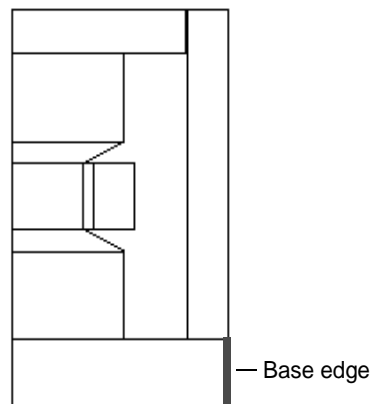
The density factor determines the length of the longest segment in relation to the shortest segment. For example, a density factor of 5 means that the longest segment is 5 times longer than the shortest segment.



6. On the Mesh toolbar, click the Logarithmic: End to Start tool . This setting places more subdivisions at the end of the edge than at the start of the edge.

## Use the uniform distribution method

### Subdivide the base

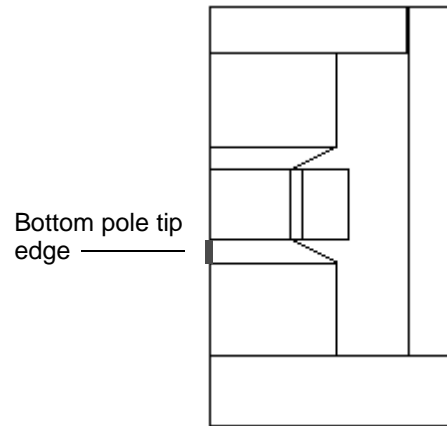
1. Click the mouse pointer on the edge of the component named Base.




2. On the Mesh toolbar, in the Number of Subdivisions text box (the leftmost text box)  , enter **5**.
3. On the Mesh toolbar, select the Uniform Subdivision tool . Five uniform subdivisions are assigned to the edge.

### Subdivide the bottom pole tip

1. Click the mouse pointer on the edge of the Bottom pole tip.  
If the entire edge of the model is selected, click again (or press TAB on your keyboard). The selection “cycles through” to the next edge.




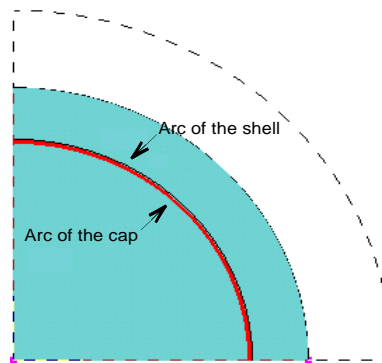
2. In the Number of Subdivisions text box, enter **3**.
3. On the Mesh toolbar, select the Uniform Subdivision tool .  
Three uniform subdivisions are added to the edge.  
The display of the mesh automatically updates.

### Subdivide the top pole tip

1. Click the mouse pointer on the edge of the Top pole tip.
2. Add 3 uniform subdivisions.
3. Click **Save**.

### Subdivide the shell and cap


1. On the Preset View toolbar, select  (positive Z axis).
2. Click the mouse pointer on the arc of the cap.  
Use the Zoom In tool to magnify the display.

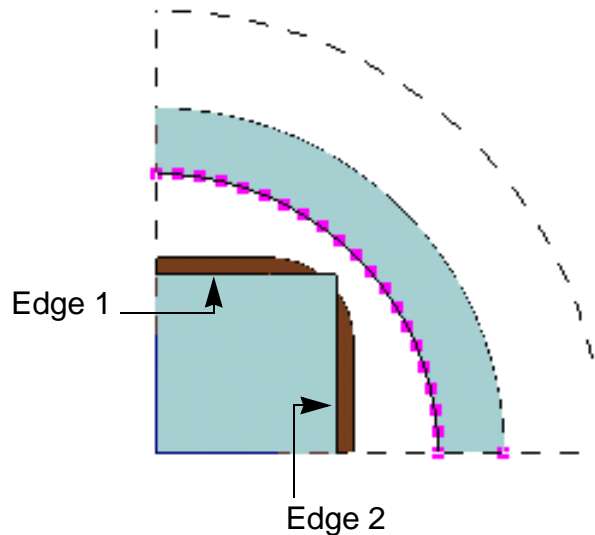


3. Add **30** Uniform subdivisions.
4. Click the mouse pointer on the inner arc of the shell that is nearest the air gap.
5. Add **30** Uniform subdivisions.


## Subdivide the top pole

In order to view the edges of the top pole, it is best to hide certain components.

1. On the Object page, select Base, the Cap, the Bottom pole, and both pole tips, right click the mouse, and then click Visible.  
The X symbol  next to the components indicates that they are hidden.




**Tip** You can select multiple edges by holding down the SHIFT or CTRL keys while selecting the edges. With the CTRL key you can de-select a selected edge by clicking on it again.

2. Select Edge 1 (as shown in the diagram above).
3. Apply 10 Uniform subdivisions.
4. Select Edge 2 (as shown in the diagram above).
5. Apply 10 Uniform subdivisions.
6. Click Save.
7. On the Object page, select Base, the Cap, the Bottom pole, and both pole tips, right click the mouse, and then click Visible.  
The check symbol  next to the components indicates that they are visible.

## View the initial mesh

After changing the subdivisions, the initial mesh can be viewed before solving. The mesh will be easier to view if the display of the air box is turned off.

1. On the Object page, select Air box, right click the mouse, and then click Visible.

The X symbol  Air box next to the Air box component indicates that it is hidden.

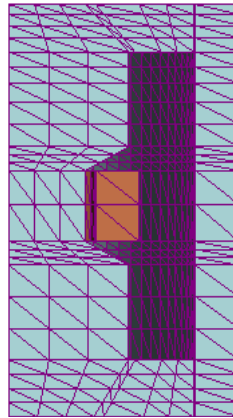
2. On the Window menu, click New Window.

A new window appears.

3. On the Preset View toolbar, select  (negative Y axis).

4. On the View menu, click Initial 3D Mesh.

The initial mesh appears in the window that you just opened and should look like the following diagram.



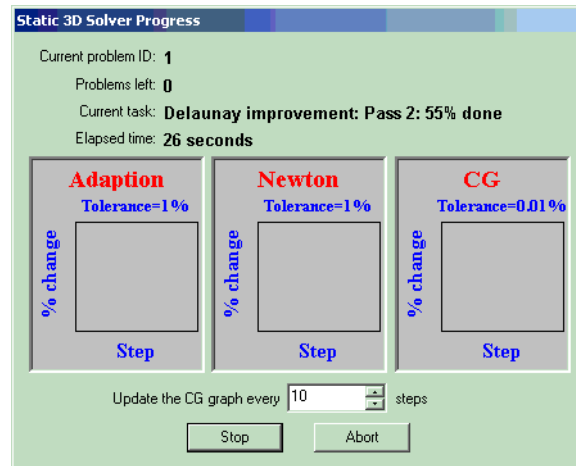
5. Close the View window that is displaying the mesh.
6. On the Object page, select Air box, right click the mouse, and then click Visible.

The check symbol  Air box next to the Air box component indicates that it is visible.

## Step 6: Solve

---

- On the Solve menu, click Static 3D.  
The Static 3D Solver Progress dialog appears.



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

## Step 7: View the solution results

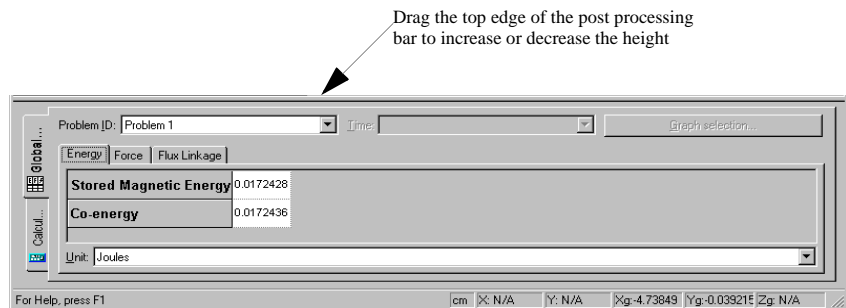
In this section, the following results are viewed:

- Stored magnetic energy
- Co-energy
- Shaded plot of  $|\mathbf{B}|$  smoothed on the surface of the model and on a slice
- Arrow plot of  $\mathbf{B}$  on the slice

### View the energy

The Post Processing bar is automatically displayed at the bottom of the Main window when the solution is complete.


**Tip** Adjust the height of the Post Processing bar by placing the cursor on its top edge, then while keeping the left mouse button pressed, dragging the bar up or down.

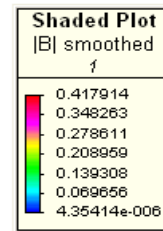
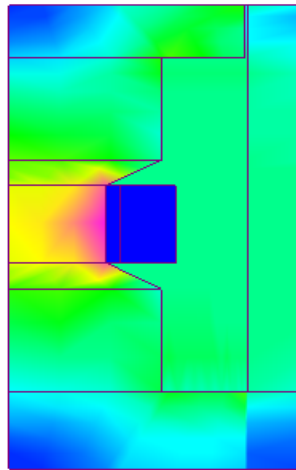


1. Click the Energy tab.  
The Energy page is displayed.
2. When you are finished with the Post Processing bar, close it by selecting Post Processing bar from the Tools menu.



## 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 X symbol  next to the Air box component indicates that it is hidden.
3. On the Project bar, select the Field tab.
4. Select the Shaded page.
5. In the Fields To Display list, make sure that  $|B|$  smoothed is selected.  
 $|B|$  smoothed is the default field.
6. Select Update View.  
The shaded plot on the surface of the model is displayed. (This may take a moment.) A color legend is displayed beside the shaded plot.

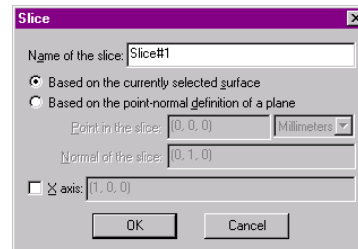


## Create a slice

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.



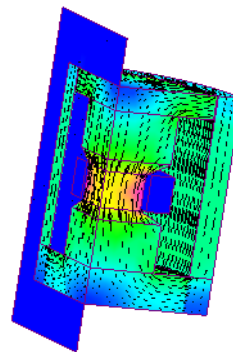
3. Make sure that Based On The Currently Selected Surface is selected.
4. Select OK to create the slice.

The shaded plot is automatically updated to display the slice.

## 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**.
3. Press Update View.

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



Rotate the model to view  
the Arrow Plot from  
different perspectives

## 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.

## 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 mesh edge subdivisions
- Viewing a shaded plot and an arrow plot



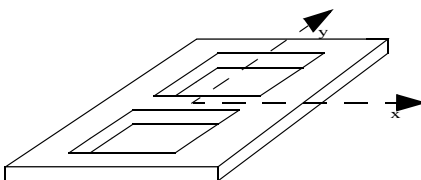
## Chapter 8

# 3D Time-harmonic tutorial: Bath plate

**Note** It is recommended that you complete the 3D “Quick Start” Tutorial in the *Getting Started Guide* before beginning the Bath plate tutorial.

## 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.

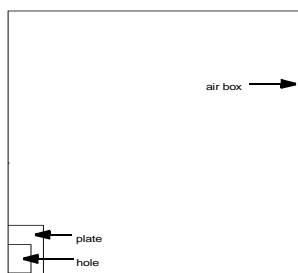


*The Bath plate with symmetry planes.*

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

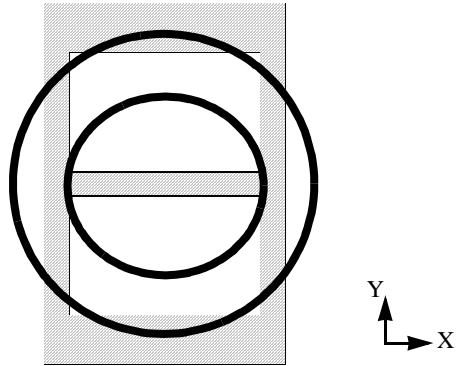
The model is symmetric about the X and Y planes, so only one-quarter of the model needs to be built and solved.

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



## 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).*

## Step 1: Open a new model

---

- Start MagNet.  
If MagNet is already running, select New from the File menu to open a new model.

### 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*.

### Set the model units

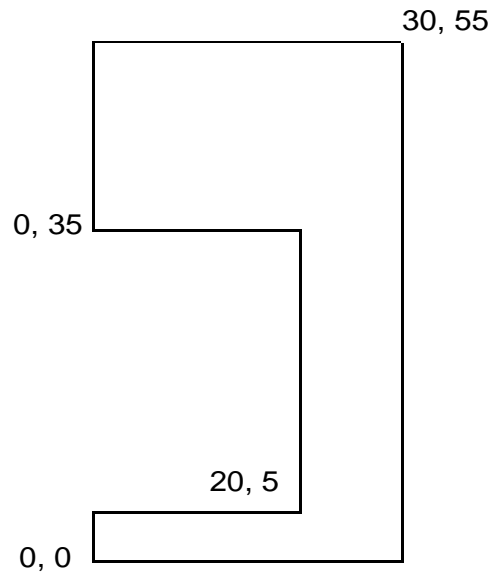
The MagNet default unit of measurement is meters. The Bath plate will be modeled in millimeters. You can set millimeters to be preferred unit of measurement in all the MagNet dialogs. This option is set in the General Model property page.

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

## Step 2: Build the geometric model



---

The Bath plate is built from one component.





### 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 Zoom toolbar, click the Update Automatically tool . This option updates the display of the model to fit inside the View window.
3. On the Draw toolbar, click the Line drawing tool .
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.

## 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 the Select Construction Slice Surface tool .
2. Click the mouse pointer inside the surface of the plate.
3. On the Model toolbar, click the Make Component in a Line tool .
4. In the Distance box, enter **6.35**.

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 Set Color and select an appropriate color*
- Description: *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**
- Relative Permittivity = **1**
- Conductivity *Siemens/m* = **0.3278e8**

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 Name box, enter **Plate**.

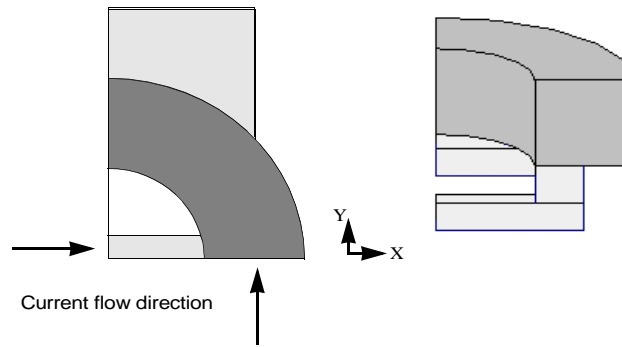
13. Click *Ok* to accept the settings.

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

## Step 3: Create the excitation

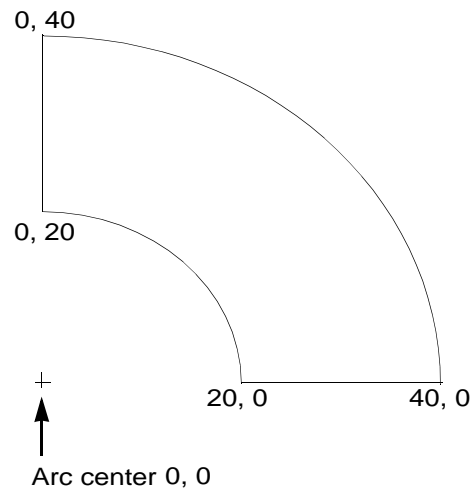
---

In this model, there is one current-driven coil. The coil is a quarter-circle 15 millimeters above the plate. Please refer to “Coil” on page 158 for more information.






### Create the coil component

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



### 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  (positive Z axis).
2. On the Object page, select Plate, Face#2 (End Face).
3. On the Draw toolbar, click the Move Construction Slice tool .
4. Make sure the option To The Currently Selected Surface is selected.
5. Click *OK*.
6. Select the Move Construction Slice tool  again.
7. In the Move Construction Slice dialog, click *Along A Line*.
8. In the Distance box, enter **15**.
9. Click *OK*.

### 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.

**Draw the geometry**

1. On the Draw toolbar, click the Arc drawing tool .
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
-----------------	-------	-------------

6. Press ESC.


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

Start coordinates            0, 20            Press ENTER


End coordinates            0, 40            Press ENTER

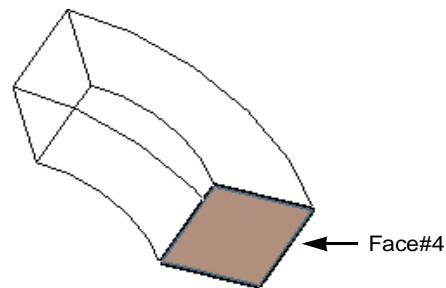
8. Press ESC.

### 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 the Make Component in a Line tool .
4. In the Distance box, enter **20**.
5. In the Material drop down list, select **Copper 5.77e7 Siemens/meter**.
6. In the Name box, enter **Current component**.
7. Click *OK* to accept the settings.
8. Click *Save*.

### Make the coil

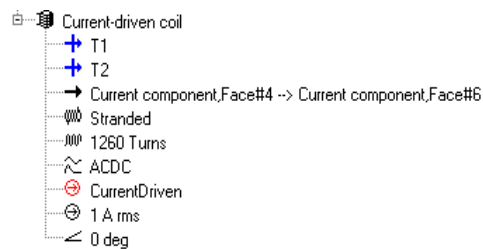
1. On the Rotation toolbar, click the Dynamic Rotation tool .
2. Using the mouse pointer, rotate the model so that the current component is in the position shown below.



3. On the Object page, select Current component, Face#4.
4. On the Model menu, click *Make Simple Coil*.  
The coil is listed in the Object page as Coil#2.

## Edit the properties of the coil

1. On the Coil page, click Coil#1.  
The Coil Attributes property page updates.
2. In the Name box, enter **Current-driven coil**.
3. In the Number of Turns box, enter **1260**.
4. In the Current per Turn box, enter **(1, 0)**.  
The current is 1 ampere per turn. The phase is 0 degrees.
5. Click *OK*.  
The Coil page is updated.

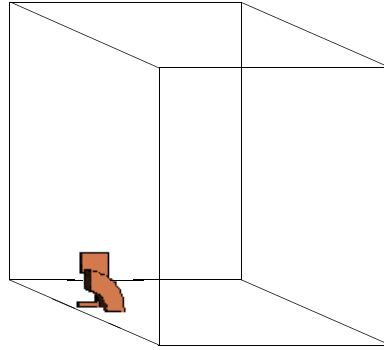


6. Click *Save*.

## Step 4: 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.



### Create the air box

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

#### Move the construction slice

1. On the Preset View toolbar, click  (positive Z axis).
2. On the Object page, select Plate, Face#1 (Start Face).
3. On the Draw toolbar, select the Move Construction Slice tool .
4. Make sure that To The Currently Selected Surface is selected.
5. Click *OK*.
6. Select the Move Construction Slice tool  again.
7. Select Along A Line.
8. In the Distance box, enter **-179.5**.
9. Click *OK*.



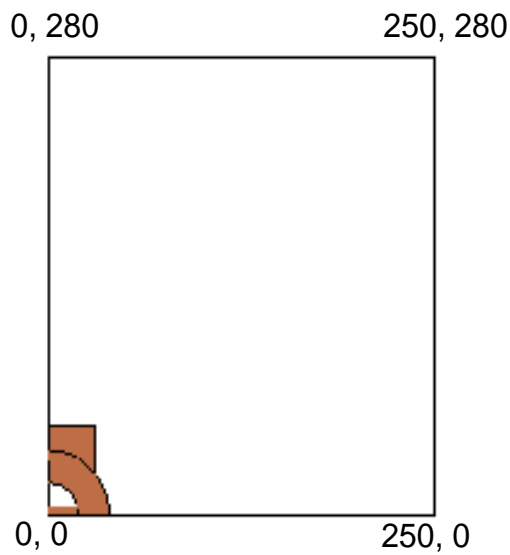
### Delete Construction Slice edges


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.

### Draw the geometry

The geometry of the air box is shown below.



1. On the Draw toolbar, click the Line drawing tool .
2. Draw the air box using the coordinates shown above.


### 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, enter **Air box**.
5. Click *Save*.

### Rotate the model

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

1. On the Rotation toolbar, click the Dynamic Rotation tool .
2. Using the mouse pointer, rotate the model.

## Step 5: Set the source frequency

---

The Bath plate is solved at 50 Hertz.

1. On the Solve menu, click *Set Solver Options*.  
The Set Solver Options dialog appears.
2. In the Source Frequency units drop down list, make sure that Hertz is selected.  
Hertz is the default unit.
3. In the Source Frequency text box, enter **50**.
4. Click OK on the dialog.
5. Click *Save*.

### Set the meshing method to extrusion

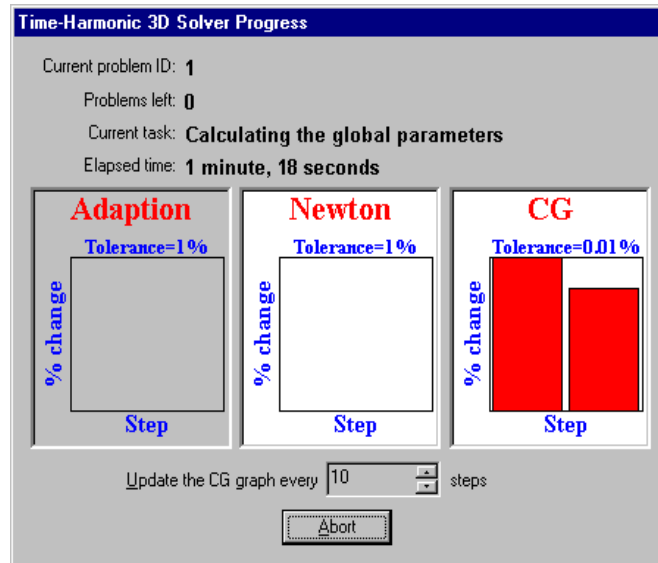
**Note:** If you do not have the Full 3D Solid Modeler license, this option is not available to you. Your default setting is extrusion.

1. From the Object page, select **Bath plate**.
2. On the Edit menu, click *Properties*.
3. Select the Mesh page.
4. In the Meshing Method To Use drop down list, select Extrusion.
5. Click *Ok*.  
You are now ready to solve the model.

## Step 6: Solve

---

- On the Solve menu, click *Time-harmonic 3D*.  
The Time-harmonic 3D Solver Progress dialog appears.



The Bath plate model takes a few seconds to solve (solution time may vary according to computer). The Solver Progress dialog automatically exits when the solution is complete.

## Step 7: View the solution results

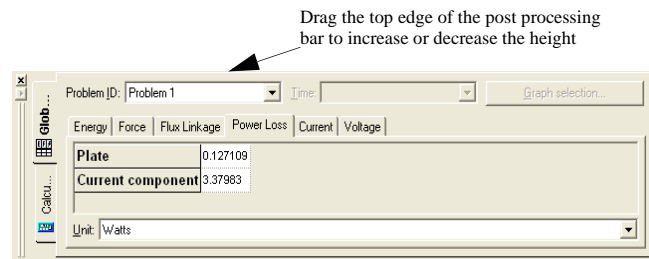
In this section, the following results are viewed:

- Time-averaged Ohmic power dissipated in each conductor.
- Plotting  $B_z$  at 0', extracted along a contour and graphed.

### View the time-averaged Ohmic power

The Post Processing bar is automatically displayed at the bottom of the Main window when the solution is complete.

**Tip** Adjust the height of the Post Processing bar by placing the cursor on its top edge, then while keeping the left mouse button pressed, dragging the bar up or down.



- Click the Power Loss tab.

The Power page displays the time-averaged Ohmic power loss in each conducting component in the model.


## Obtain $B_z$ at 0' and graph the results

To obtain  $B_z$  at 0' for graphing, the following steps must be completed:

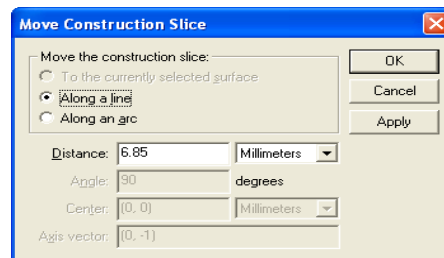
- Move construction slice.
- Update the field view to display B at 0'.
- Plot  $B_z$  at 0' on a graph.

### 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. On the Draw toolbar, select the Move Construction Slice tool .

The Move Construction Slice dialog appears.



2. Make sure *Along A Line* is selected.
3. In the Distance box, enter **6.85** Millimeters.
4. Click *OK*.

### Update field view to display B at 0'

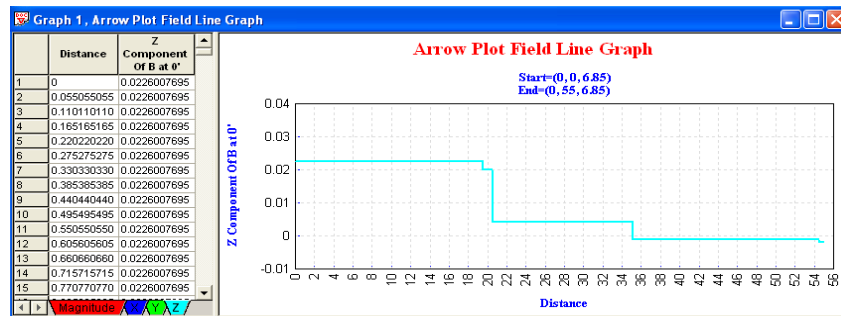
1. On the Project bar, select the Field tab.
2. For both the Contour and Shaded plots (click the tabs at the bottom of the Field page), in the Fields To Display list, select **None**.
3. Click the Arrow plot tab, and then in the Fields To Display list, select **B at 0'**.
4. At the bottom of the Field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

## Plotting $B_z$ at 0' on a graph

In this procedure we are going to use the 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.



## 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*.

## 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 power.
- Obtaining and creating a graph of  $B_z$  at 0' along the contour

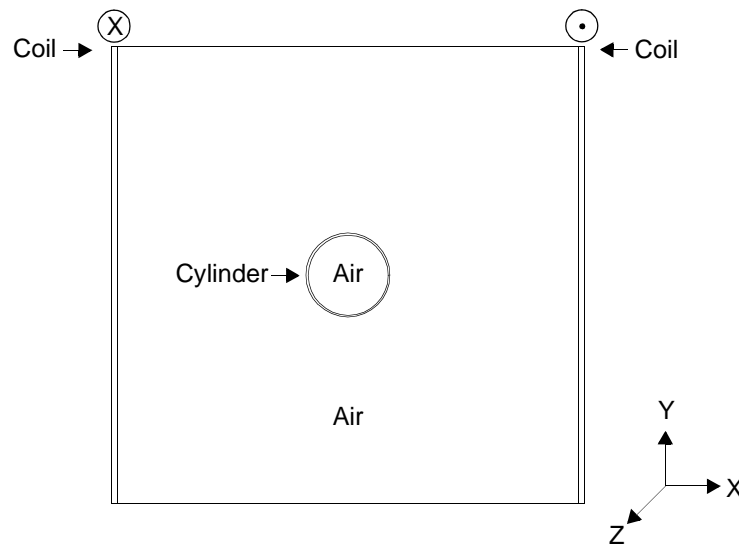


## Chapter 9

# 3D Transient Tutorial: Felix short cylinder

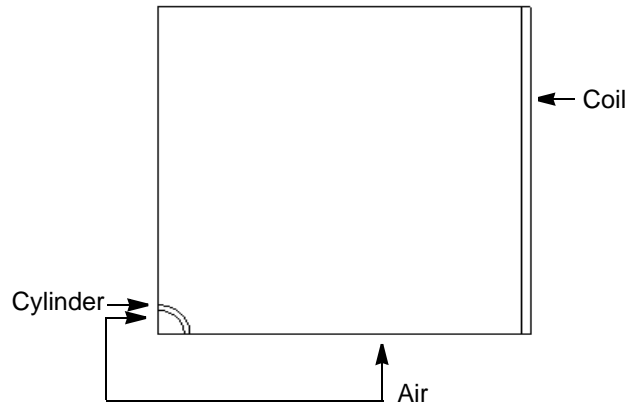
### Modeling plan

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

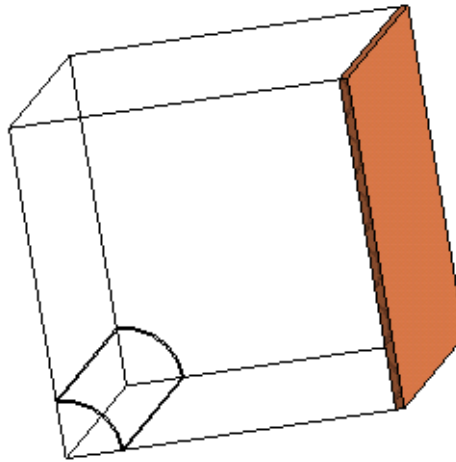


After solving, the shaded plot is viewed and animated. Once the animation is complete, the stored energy of the system is graphed. Both the animation and the graph are viewed through all the time steps.

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.

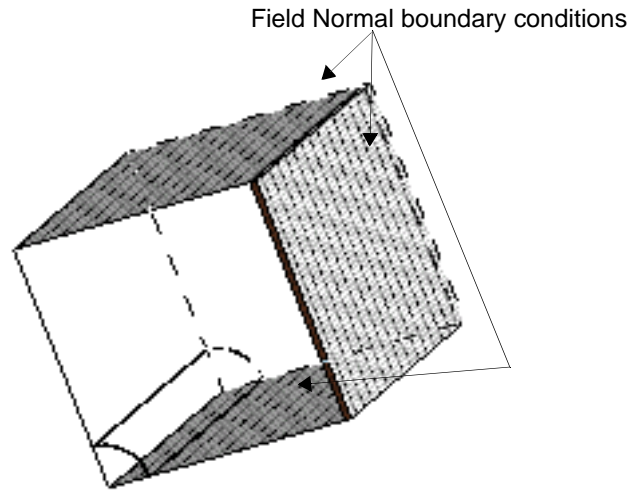


After drawing, the geometry is swept into components and a coil is created from one of the components.

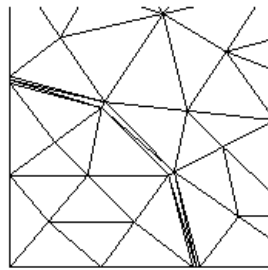


**Tip** The Field Normal boundary condition on the right surface of the air box (representing the outside of the coil) forces the coil flux outside to infinity. If the surface had the Flux Tangential boundary condition, the coil flux would be forced to return inside the coil (which would give incorrect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

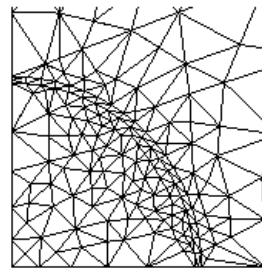
Boundary conditions are applied to indicate symmetry. The Field Normal boundary condition is applied to the top, bottom, and right surfaces of the air space. The default boundary condition, Flux Tangential, is automatically applied to the remaining surfaces.



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




Default initial mesh



Modified initial mesh

## Step 1: Open a new model

---

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

### 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.

### Set the preferred unit for time

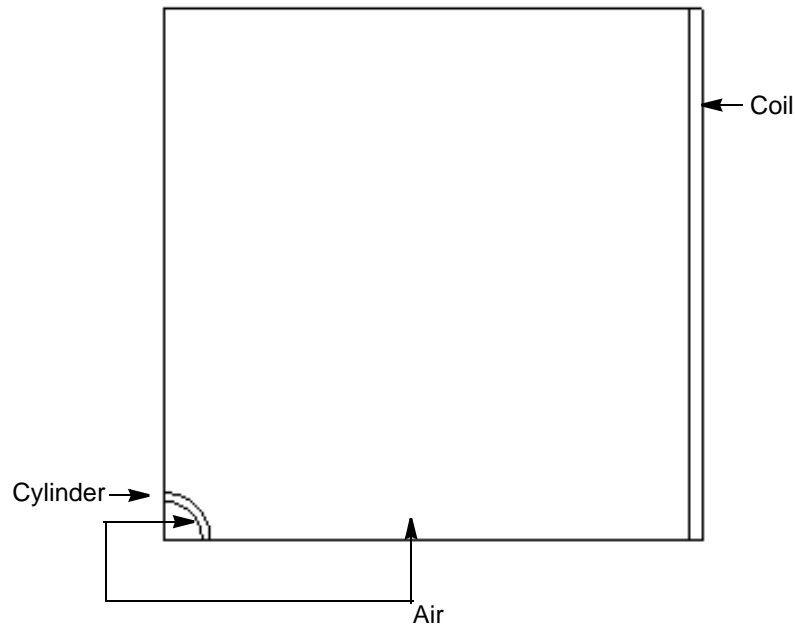
The MagNet default unit for time is milliseconds. The Felix short cylinder time steps will be measured in seconds. You can set seconds to be preferred unit for time in all the MagNet dialogs. This option is set in the Model property page.

1. In the Object page of the Project bar, select the name of the model (in this case, 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.

## Step 2: Build the geometric model


---

The geometry of the model is shown below.



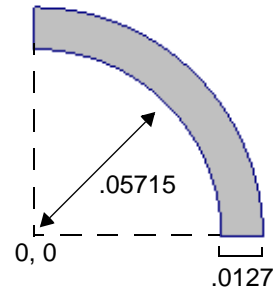
### Set the drawing area

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 MagNet default).

1. Turn off the display of the Construction Grid (if it is displayed).  
If the grid is visible, toggle Construction Grid on the View menu.
2. On the Zoom toolbar, click the Update Automatically tool .  
This option updates the display of the model to fit inside the View window.

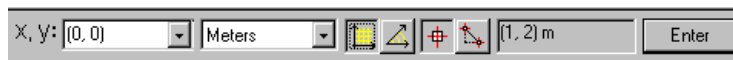
## Create the cylinder component




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



### 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 the Arc drawing tool .
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 the Line drawing tool .

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

8. Press ESC.

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

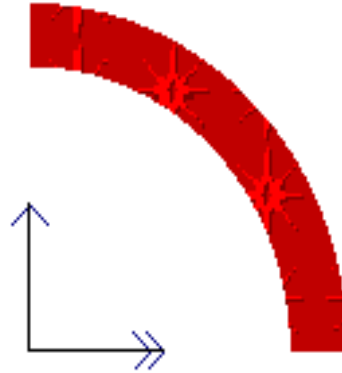
Start coordinates      0.05715, 0      Press ENTER


End coordinates      0.06985, 0      Press ENTER

10. Press ESC.

### 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 the Make Component in a Line tool .
4. In the Distance box, enter **0.1**.
5. In the Name box, enter **Cylinder**.

**Note** If you have already created the material called Aluminum 6061 in the 2D Transient tutorial, please proceed to step 12. If not, proceed to step 6.

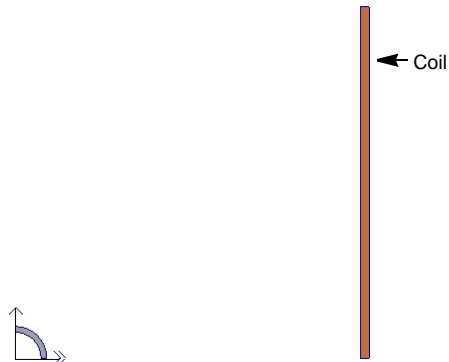
6. Click New Material.  
For this problem, you will have to create a new material in your material database.




7. On the General page, enter the following data:
  - Name: **Aluminum 6061**
  - Display color: *Click Set Color and select an appropriate color*
  - Description: *Optional*
8. Click Next.
9. On the Options page, select the following:
  - Magnetic ***Permeability***
  - Electric ***Conductivity***
10. 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**
11. Once you have entered all the values, click Finish in the Confirmation page to create the new material.
12. From the Material drop down list, select **Aluminum 6061**.
13. Click Ok to accept the settings.
14. On the File menu, click Save.

## Create the coil component

The geometry of the coil component is shown below.



### Draw the coil component

1. On the Draw toolbar, click the Line drawing tool .
2. 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
-----------------	--------	-------------

3. Press ESC.

### 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 the Make Component in a Line tool .
4. In the Distance box, enter **0.2**.
5. In the Material drop down list, select **Copper: 5.77e7 Siemens/meter**.
6. In the Name box, enter **Coil component**.
7. Click OK to accept the settings.
8. On the File menu, click Save.

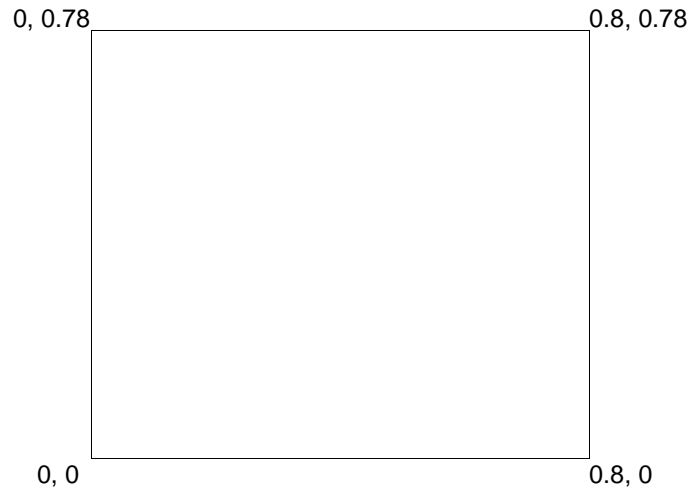
### Delete Construction Slice edges

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.

## Create the air box component

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




### Draw the geometry of the air box


1. On the Draw toolbar, click the Line Drawing tool .
2. 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
3. Press ESC to stop drawing.

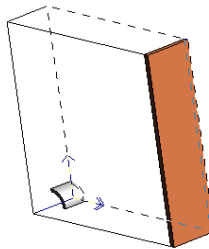
### Make the component of the air box

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 the Make Component in a Line tool .
4. In the Distance box, enter **0.2**.
5. In the Material drop down list, select **AIR**.
6. In the Name box, enter **Air box**.
7. Click OK to accept the settings.
8. On the File menu, click Save.

### Rotate the display of the model

1. On the Preset View toolbar, click the Dynamic Rotation tool .
2. Click the mouse pointer on the 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.

The display is rotated about the center of the model. The model is displayed as a wireframe during the rotation.



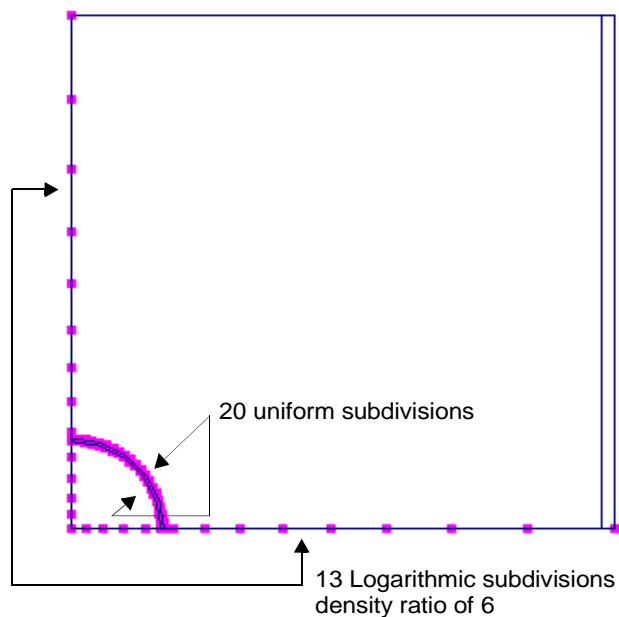
## Step 3: Modify the mesh

---

In the finite element method of analysis, the model is divided into a mesh of tetrahedral-shaped elements. The 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 subdivide component edges into segments. The number of edge segments corresponds to the number of elements along the edge. Note that the subdivisions propagate throughout the entire model in the sweep direction.

The edge subdivisions that are modified in this model are shown in the diagram below.




## Set the meshing method to extrusion

**Note:** If you do not have the Full 3D Solid Modeler license, this option is not available to you. Your default setting is extrusion.

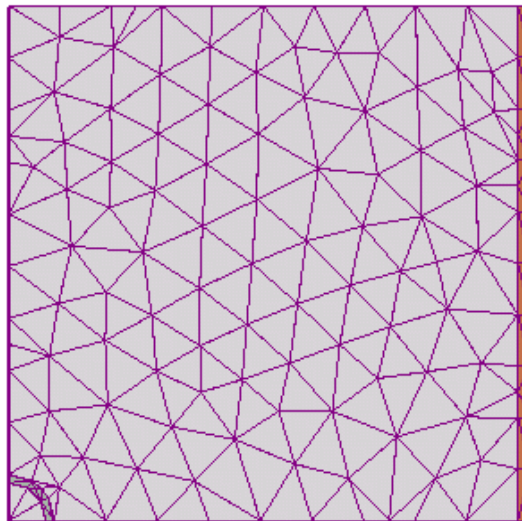
1. From the Object page, select **Felix short cylinder**.
2. On the Edit menu, click Properties.
3. Select the Mesh page.
4. In the Meshing Method To Use drop down list, select Extrusion.
5. Click Ok.

## View the initial mesh

Before changing the subdivisions, the default initial mesh can be viewed.

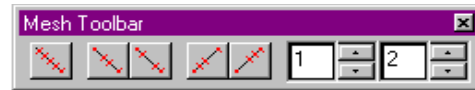
1. On the Preset View toolbar, click  (positive Z axis).
2. On the View menu, click Initial 2D Mesh.

The initial mesh appears in the View window. The mesh should look like the following diagram.



## Modify the edge subdivisions

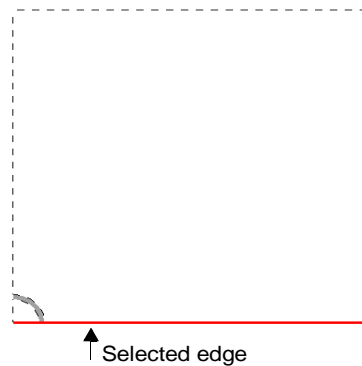
Subdivisions can be applied to an edge using either a logarithmic or uniform (linear) scale. Note that subdivisions are edited on the model, not directly on the mesh. Subdivisions are applied to the model using the Mesh toolbar. Please see the Help for instructions on displaying toolbars.



Mesh toolbar: Floating position

### Subdivide the edges of the air box

1. On the View menu, click Solid Model.
2. On the Edit menu, click Select Component Edges.
3. Click the mouse pointer on the bottom edge of the air box (as shown in the following diagram).




4. On the Mesh toolbar, in the Number of Subdivisions text box (the left text box)  , enter **13**.
5. In the Subdivision Density Factor text box (the right text box), enter **6**. The toolbar should look like the diagram below.




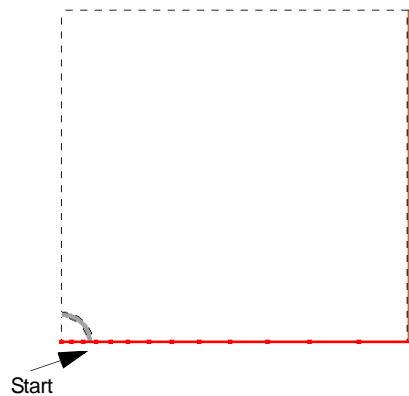


The density factor determines the length of the longest segment in relation to the shortest segment. For example, a density factor of 6 means that the longest segment is 6 times longer than the shortest segment.

6. On the Mesh toolbar, click the Logarithmic: Concentrated at Start tool .


This setting places more subdivisions at the start of the edge than at the end of the edge. The subdivisions should look like the following diagram.

**Tip** If the subdivisions are concentrated at the other end of the edge, click the Logarithmic: Concentrated at End tool .




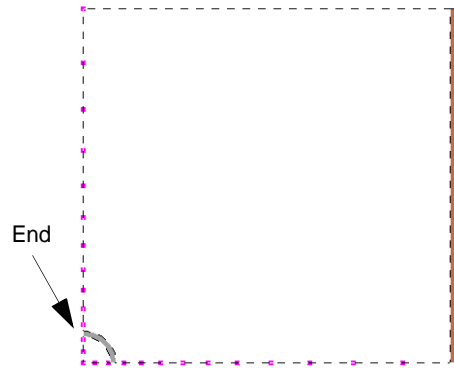
7. Click the mouse pointer on the left edge of the air box as shown below.



8. On the Mesh toolbar, click the Logarithmic: Concentrated at End tool .

This setting places more subdivisions at the end of the edge than at the start of the edge. The subdivisions should look like the following diagram.

**Tip** If the subdivisions are concentrated at the wrong side of the edge, click the Logarithmic: Concentrated at Start tool .

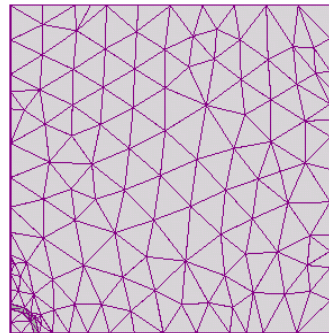


9. On the File menu, click Save.

### View the changes to the mesh


- On the View menu, click Initial 2D Mesh.  
The mesh updates (this may take a moment).

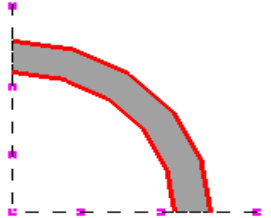
The mesh should look like the following diagram.





## Subdivide the edges of the cylinder

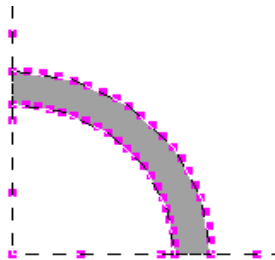
**Tip** Holding the SHIFT or CTRL keys while selecting objects enables you to select multiple objects. Using the CTRL key enables you to select an object a second time to de-select it.

1. On the View menu, click Solid Model.
2. Use the Zoom In tool  to enlarge the area around the cylinder.
3. On the Edit menu, click Select Component Edges.
4. While holding down the SHIFT key on your keyboard, click the mouse pointer on each of the arcs of the cylinder.  
The arcs are highlighted.




5. On the Mesh toolbar, in the Number of Subdivisions text box (the left text box) , enter **20**.  
The density ratio does not apply to uniform subdivisions.
  6. On the Mesh toolbar, click the Uniform Edge Subdivision tool .
- 20 uniform subdivisions are assigned to each of the edges.

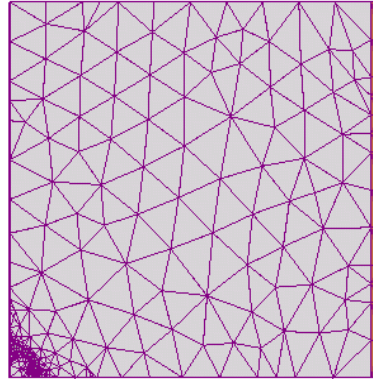
**Tip** To improve the display of arcs, you can decrease the smoothness angle using the SetCurveSmoothnessAngle scripting command. Please see the Help for more information.




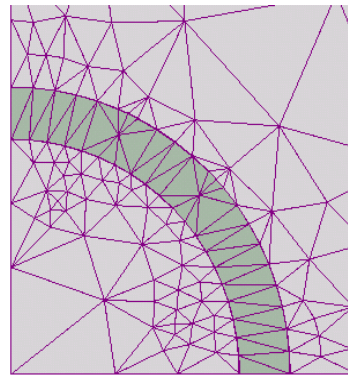
**View the changes to the mesh**

1. On the Zoom toolbar, click the Update Automatically tool .
  2. On the View menu, click Initial 2D Mesh.
- The mesh updates (this may take a moment).

The mesh should look like the following diagram.



3. Use the Zoom In tool  to enlarge the area around the cylinder.
- The mesh should look like the following diagram.



## Step 4: Define boundary conditions



---

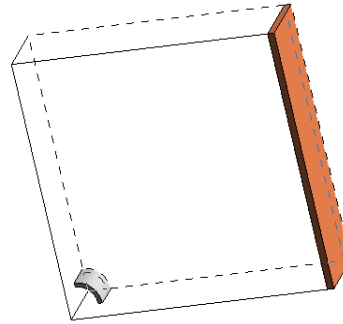
The Field Normal boundary condition is applied to three surfaces of the air box: the top, bottom, and right surfaces. The default boundary condition, Flux Tangential, is automatically applied to the remaining surfaces.

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 (along side) the boundary.

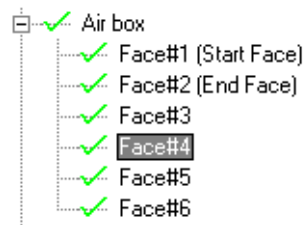
### Apply the Field Normal boundary condition

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

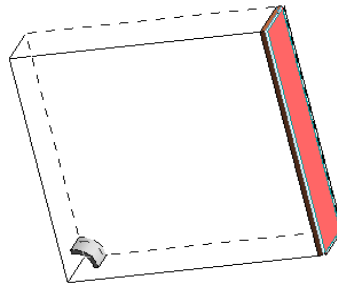
1. On the View menu, click Solid Model.
2. On the Zoom toolbar, click the Update Automatically tool .
3. On the Preset View toolbar, click the Dynamic Rotation tool .
4. Rotate the model to a 3D view (similar to the diagram below). This rotation displays the surfaces to which the Field Normal boundary condition will be applied.



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



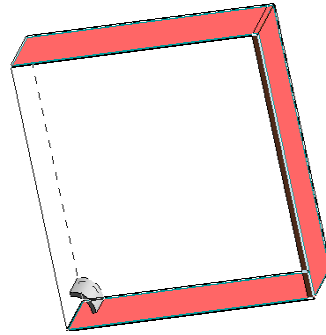
The surface is selected in the View window.




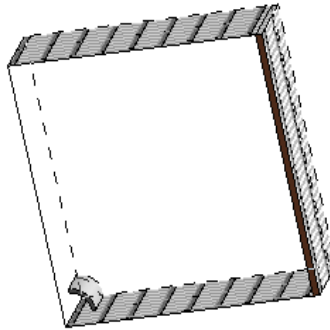
7. While holding down the CTRL key on your computer keyboard, also select Air box, Face#3 and Face#5 .

**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.

The three surfaces are selected in the View window.



8. On the Boundary Condition toolbar, click Field Normal .
9. The Field Normal boundary condition is applied to the selected surfaces.

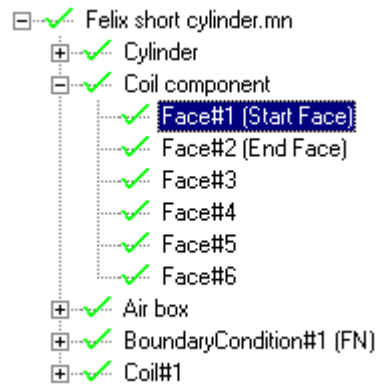


10. On the File menu, click Save.

## Step 5: Create the coil

---

1. On the Object page, select Coil component, Face#1 (Start Face).  
The current flow follows the sweep direction of the 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.

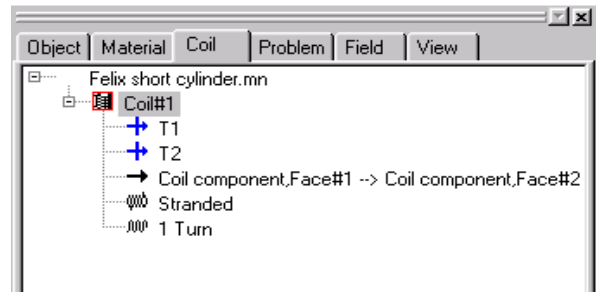


## Step 6: Edit the coil properties

---

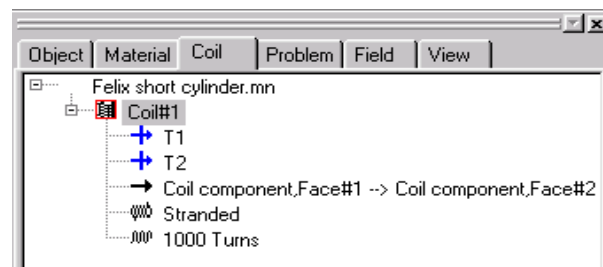
1. On the Project bar, click the Coil tab.

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. In the Number Of Turns box, enter **1000**, then press Tab.
5. Click OK.

The coil page is automatically updated.

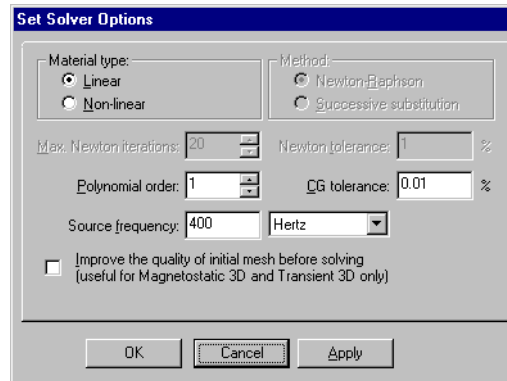


6. On the File menu, click Save.

## Step 7: Set the source frequency

---

1. On the Solve menu, click Set Solver Options.  
The Set Solver Options dialog appears.



**Tip** If the Linear option is not set, the solver will automatically detect the presence of linear materials.

2. Select the Linear option.
3. In the Source Frequency box, enter **400**.  
The default unit is Hertz.
4. Click OK.

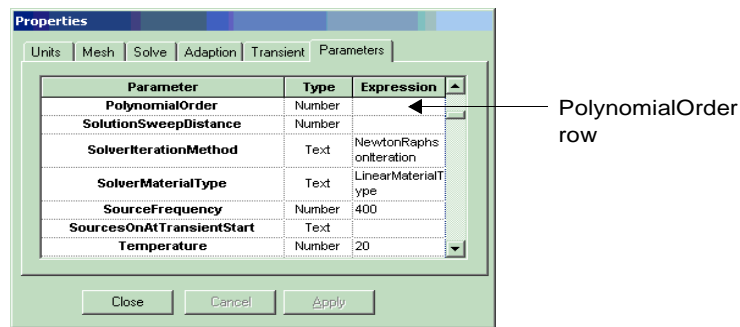
## Step 8: Change the polynomial order

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.

The polynomial order is set in the Model property page.

### To change the polynomial order

1. On the Object page, select Felix Short Cylinder.mn.
2. On the Edit menu, click Properties.  
The Model property page appears.
3. Click the Parameters tab.



4. Scroll down the list of parameters until the PolynomialOrder row appears.
5. In the Expression column, enter 2, then press Tab.

PolynomialOrder	Number	2
-----------------	--------	---

6. Click OK.
7. On the File menu, click Save.

## Step 9: 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.



Circuit Components toolbar



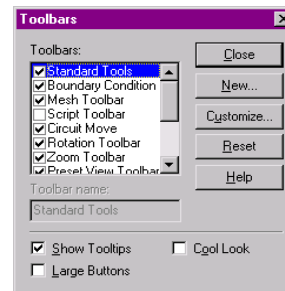
Circuit Move toolbar



Circuit Alignment toolbar

### Display the Circuit toolbars

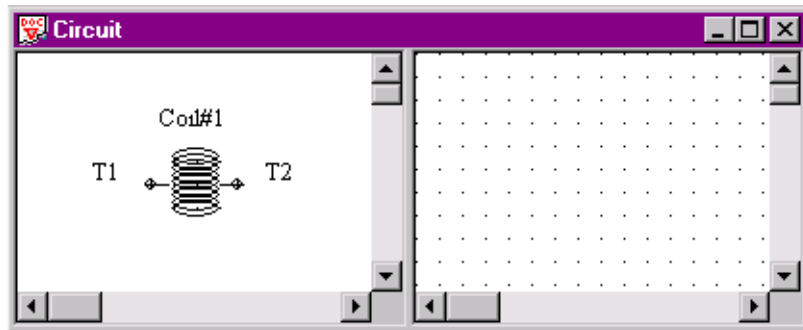
1. On the Tools menu, click Toolbars.  
The Toolbars dialog is displayed.




2. In the Toolbars list, select the following toolbars:
  - Circuit Move
  - Circuit Alignment
  - Circuit Components
3. On the Toolbars dialog, click Close.  
The toolbars are displayed.

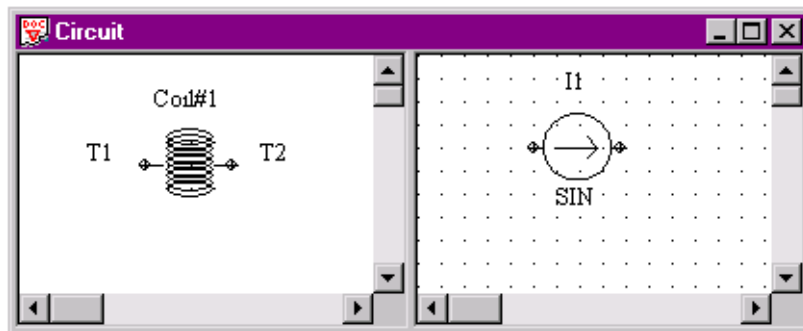
## Create the circuit


1. On the Circuit menu, click New Circuit window.  
A Circuit window opens. The left pane of the Circuit window displays the available coils in the model.

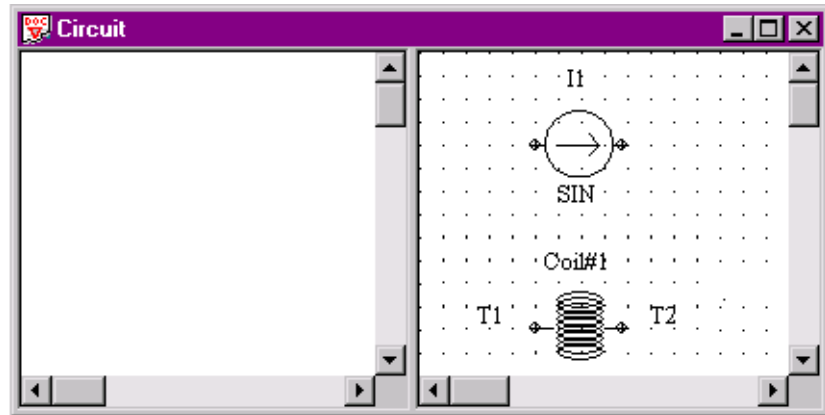




**Tip** The Circuit tools are also available on the Circuit menu. For example, to add a current source to the circuit, select Current Source on the Circuit menu.

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 highlighted 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.



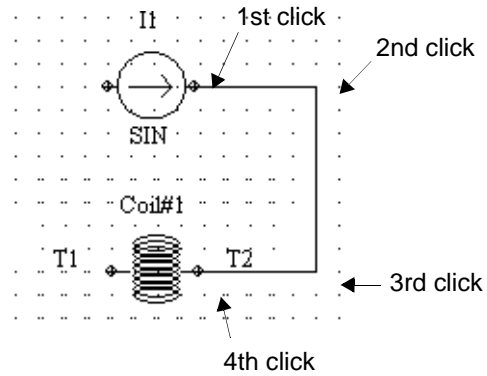
4. On the Selection toolbar, click the Select tool .
5. Select the coil in the left pane of the window, and then drag the coil to the right pane.
6. If necessary, re-size the window by dragging on an edge of the window.



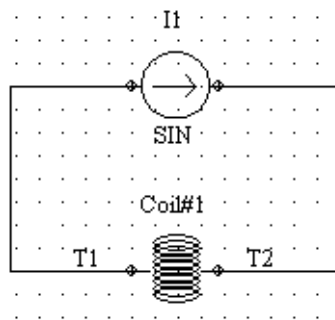
7. Select the coil and the current source with the mouse pointer. Press the SHIFT key on your keyboard while selecting the components.
8. On the Circuit Align toolbar, click Align Center .  
The components are aligned at the position of the last selected component.
9. On the Circuit Components toolbar, click Connection .  
The Connection tool is used to add 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.



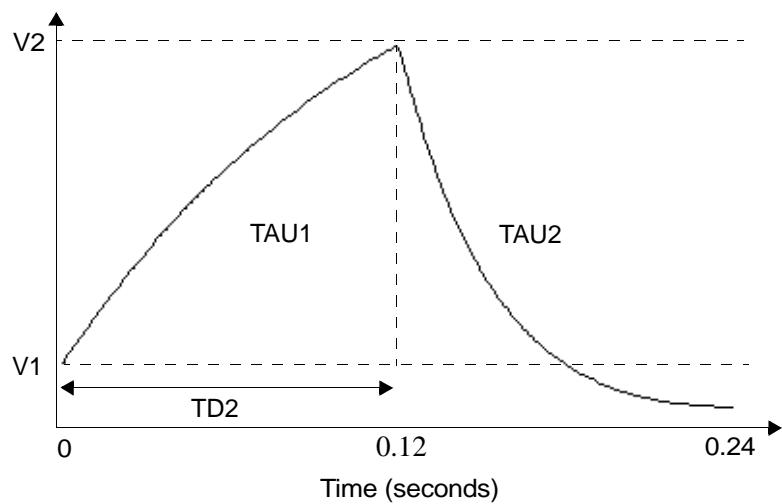
11. End the connection on the right terminal of the coil.
12. Draw a second connection as shown in the following diagram.



13. On the File menu, click Save.

# Step 10: Edit the waveform properties

The exponential waveform of the source looks like the following diagram:



The properties of the waveform are as follows:

<b>V1</b>	0	amperes
<b>V2</b>	98.19	seconds
<b>TD1</b>	0	seconds
<b>TAU1</b>	0.12	seconds
<b>TD2</b>	0.12	seconds
<b>TAU2</b>	0.0397	seconds
<b>START</b>	0.12	seconds
<b>STOP</b>	0.24	seconds
<b>STEP</b>	0.005	seconds



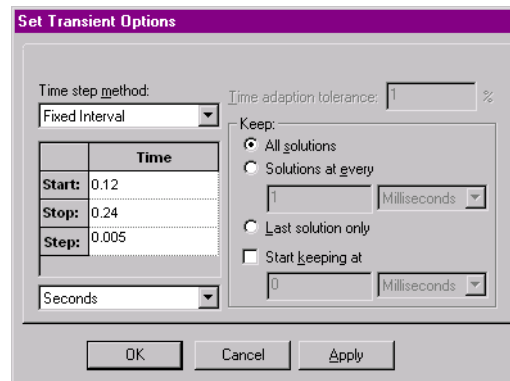
## Set the start, stop and step times

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.

**Note** The transient solver assumes the source values before the start time are equal to the values at the start time **only if** the parameter SourcesOnAtTransientStart is set to Yes.


1. From the Object page, select the model (i.e., Felix Long Cylinder.mn).
  2. On the Edit menu, click Properties.
  3. Select the Parameters page.
  4. Scroll down to the parameter **SourcesOnAtTransientStart** and type **Yes** in the Expression column.
  5. Click OK.
  6. On the Solve menu, click Set Transient Options.
- The Set Transient Options dialog appears.

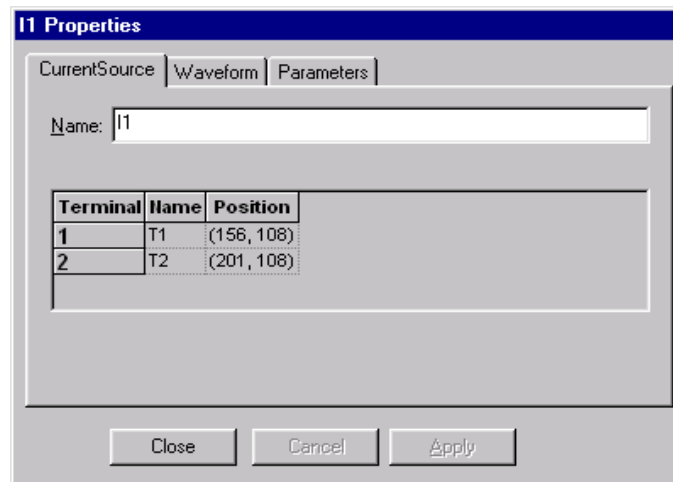


7. Make sure that *Fixed Interval* is selected as the Time Step method and *Seconds* as the unit for time, and then make the following modifications for Time:
  - Start = **0.12** Seconds
  - Stop = **0.24** Seconds
  - Step = **0.005** Seconds
8. Click OK.

## Edit the waveform

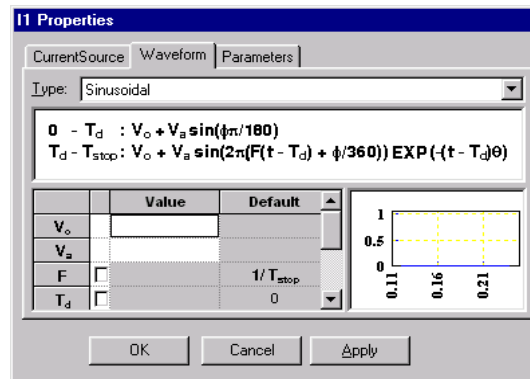
### Open the waveform editor

1. On the Selection toolbar, click the Select tool .
  2. In the right pane of the Circuit window, right-click the current source.
  3. On the pop-up menu, click Properties.
- The I1 Properties page appears.

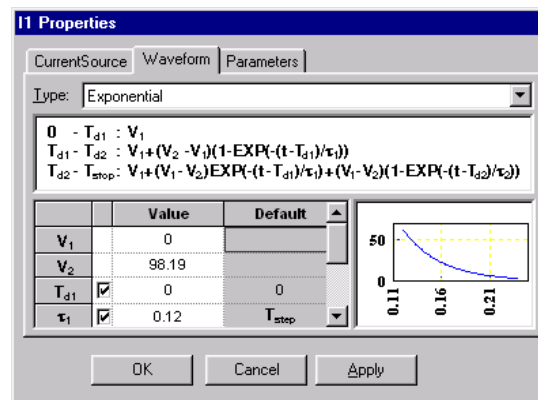


4. On the I1 Properties page, click the Waveform tab.  
The Waveform property page is displayed.

## Edit the waveform

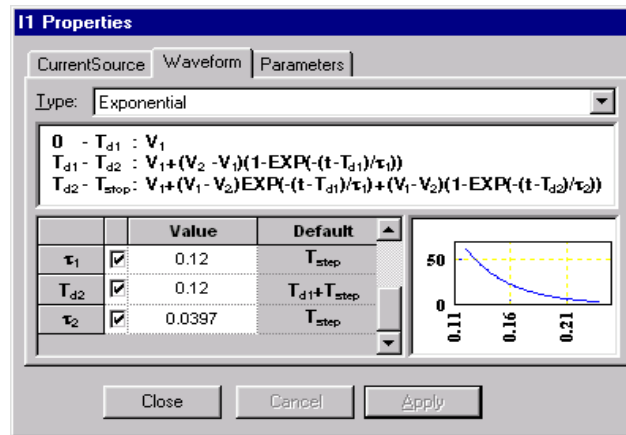


1. In the Type drop down list, select **Exponential**.  
The property page updates.



2. In the Waveform page, make the following modifications:
  - In the V1 text box, type **0**.
  - In the V2 text box, type **98.19**.
  - For T<sub>d1</sub>, click the check box, and then type **0**.
  - For tau<sub>1</sub>, click the check box, and then type **0.12**.
  - For T<sub>d2</sub>, click the check box, and then type **0.12**.
  - For tau<sub>2</sub>, click the check box, and then type **0.0397**.
3. Click Apply.

The waveform is displayed in the property page. Note that the waveform is displayed from the start time of 0.12 seconds.

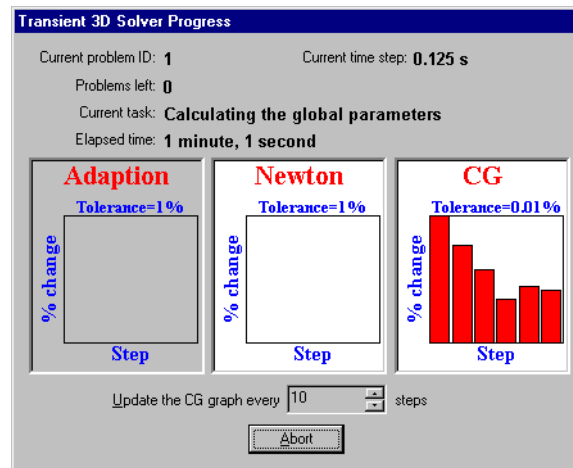


4. Click Close.
5. On the File menu, click Save.

## Step 11: Solve

---

- On the Solve menu, click Transient 3D.  
The Transient 3D Solver Progress dialog appears.



The cylindrical shield takes less than 4-5 minutes to solve (solving time may vary according to computer). The Solver Progress dialog automatically exits when the solution is complete.

## Step 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 power at the last time step of each conducting component
- A graph of the instantaneous power 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.

### Create a new slice


1. From the Tools menu, click New Slice.
2. 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)**
3. Click Ok.

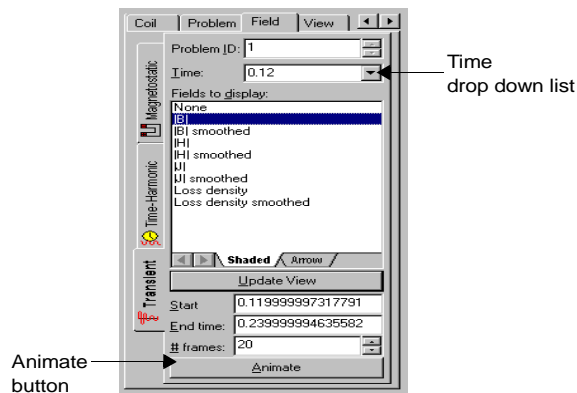
### Hide components

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 X appearing next to each component object indicates that both are hidden.

## 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. (Use the arrow on the Project bar  to scroll the pages if necessary.)  
The Field page opens.





2. Click the shaded plot tab (at the bottom of the Field page).
3. In the Fields To Display list, select  $|B|$  smoothed.
4. In the Time drop down list, make sure 0.12 is selected (the first time step).
5. At the bottom of the Field page, press Update View.  
The shaded plot is displayed. (This may take a moment.)

**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.

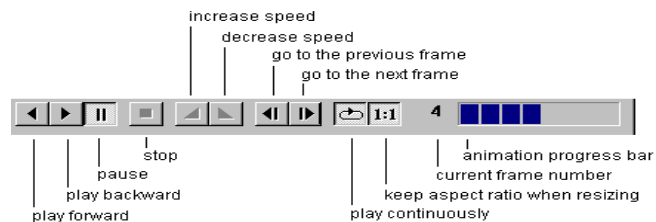


## Animate the shaded plot


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. Use the Zoom In tool  to enlarge the area around the cylinder.
4. On the Field page, select the 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 the Stop button  when you are finished viewing the animation.



## Save the animation

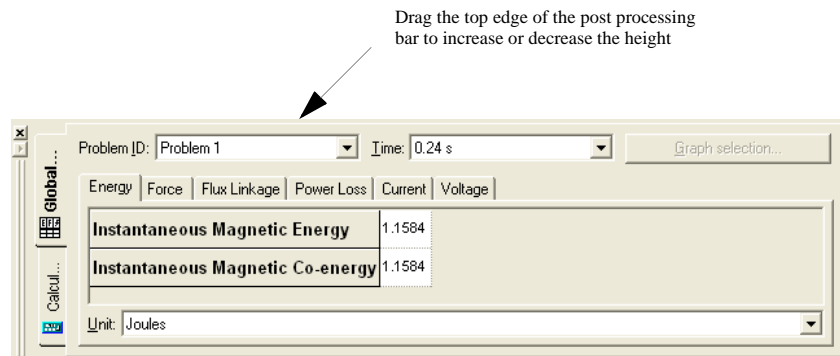
1. On the File menu, click Save.  
The Save As dialog box appears.
2. In the File Name text box, enter **Felix short cylinder**.  
The animation extension .ban is automatically added.
3. On the File menu, click Save.  
The animation is saved.
4. On the File menu, click Close.  
The Animation window closes.

## View the stored energy

The Post Processing bar is automatically displayed at the bottom of the Main window when the solution is complete.

1. On the Post Processing bar, click the Global Quantities tab.
2. From the Time drop down list, select 0.24s.
3. Click the Energy tab.  
The Energy page of the Post Processing bar displays the stored energy in the system.

**Tip** Adjust the height of the Post Processing bar by placing the cursor on its top edge, then while keeping the left mouse button pressed, dragging the bar up or down.



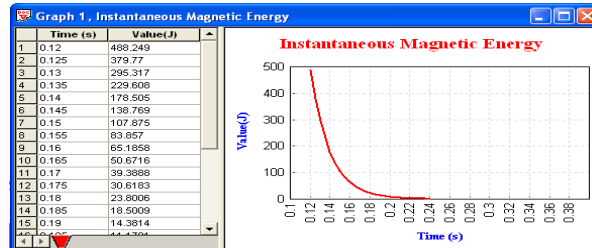
## Graph the energy across time

1. Click the mouse pointer in the Instantaneous Magnetic Energy value.



2. Press the Graph Selection button.

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



## Save the model

You have now completed the Felix short cylinder.

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

## Summary

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

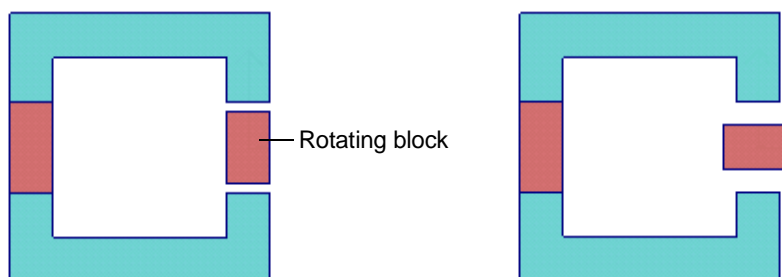
- Creating a new material
- Creating a circuit
- Defining an exponential waveform
- Viewing a shaded plot
- Animating a shaded plot
- Viewing the stored energy in the system
- Graphing the power across time.

## Chapter 10

# Parameterization tutorial: C-core with a rotating block

### Modeling plan

This tutorial builds and solves a C-core with a rotating block. Using 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.

**Note** If you have created the C-core with a permanent magnet in the *Getting Started Guide*, you can save the model to a new name (C-core with rotating block.mn) and then add the rotating block following the instructions on page 225. You can then proceed to Step 4 on page 230.

## Step 1: Open a new model

---

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

### 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.

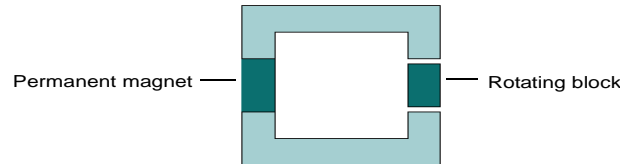
### Set the maximum element size

1. In the Object page of the Project bar, select the name of the model (in this case, C-core with a rotating block).
2. On the Edit menu, click Properties.  
The Properties dialog appears.
3. Select the Mesh tab.
4. In the “Specify meshing control options” box, click inside the Maximum element size checkbox, and then type **1** in the text box.
5. Click OK.

## Step 2: 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.



### 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.





Draw toolbar

### 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.

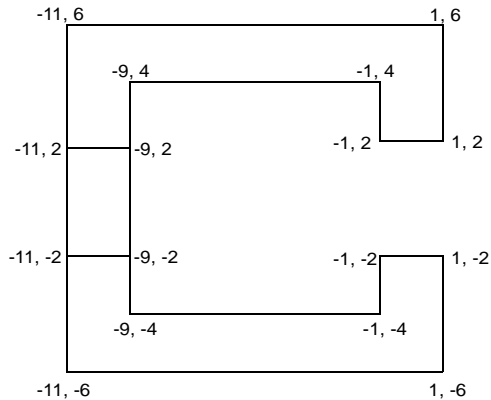
1. Make sure the Construction Grid is displayed.  
If the grid is not visible, select Construction Grid on the View menu.
2. On the View menu, click Set Construction Grid.  
The Set Construction Grid dialog appears.
3. On the Set Construction Grid dialog, enter the following:
  - Minimum X and Y text boxes: **-20**.
  - Maximum X and Y text boxes: **20**.


**Tip** You can magnify the grid with the Zoom In tool . On the Zoom toolbar, click , then click the mouse pointer in the View window. The grid is enlarged.

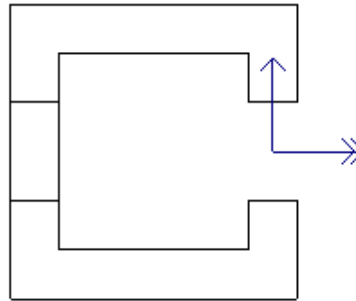
4. Click OK on the Set Construction Grid dialog.
5. On the Draw menu, make sure Snap to Grid is selected.

### Draw the geometry

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



1. On the Draw toolbar, select the Line drawing tool .
  2. Using the mouse pointer, place the coordinates using the above diagram as a guide.
  3. To stop drawing, press ESC on your computer keyboard.
- When you are finished drawing the C-core, the geometry should look like the following diagram.

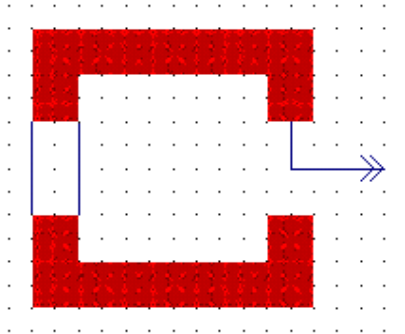



## Make the components of the C-core

Three-dimensional components can now be made from the surfaces of the C-core.

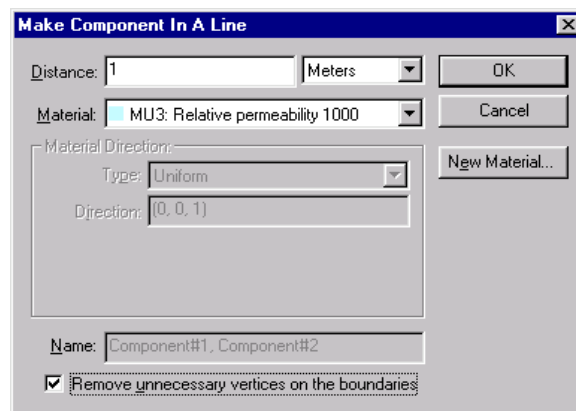
1. On the Edit menu, click Select Construction Slice Surfaces.
2. Hold down the Shift key and click the mouse pointer inside each of the surfaces of the C-core.

The surfaces are highlighted when selected.



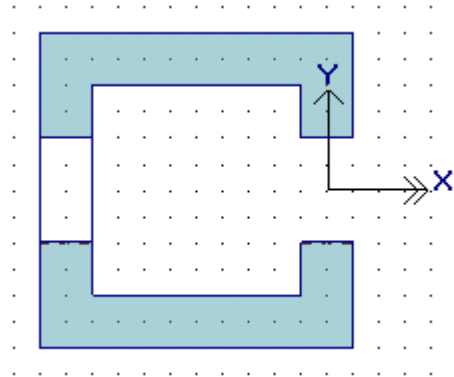
3. On the Model toolbar, click the Make Component in a Line tool .

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, select Save.

The components are created.



8. Rename the components, using the Properties page:  
 Component#1 to **Core upper half**  
 Component#2 to **Core lower half**

## Make the component of the permanent magnet

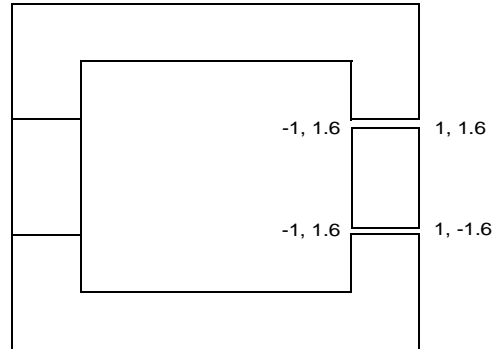
1. Click the mouse pointer inside the construction surface of the permanent magnet.
2. On the Model toolbar, click the Make Component in a Line tool .
3. In the Material drop down list, select **PM10: Brem 1.0 mur 1.0**.
4. In the Material Direction Type drop down list, select **Uniform**.
5. In the Direction text box, enter **(0, 1, 0)**.
6. In the Name box, enter **Permanent Magnet**.
7. Click OK to accept the settings.
8. Click Save.






## Create the rotating block

### Draw the geometry

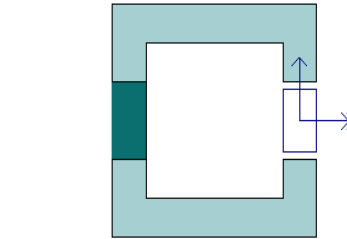
The geometry of the rotating block is now added to the C-core.




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

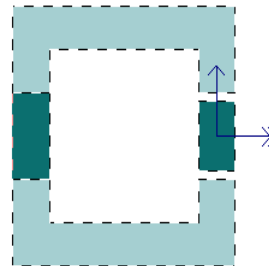


### Make the component of the rotating block

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 the Make Component in a Line tool .

The Make Component in a Line dialog box appears.

4. In the Distance box, enter **1**.
5. In the Material drop down list, select **MU3: Relative permeability 1000**.
6. In the Name text box, enter **Rotor**.
7. Click OK to accept the settings.  
The component is created.
8. Click Save.



## Step 3: Add a boundary condition

---

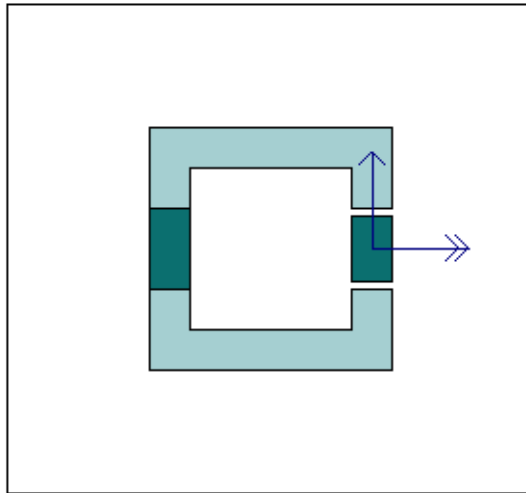
Each model needs a definition of where its boundary in space exists and how the magnetic flux will flow in relation to this boundary.

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.

### Create the air box

The C-core is centered in the air box.




## 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.

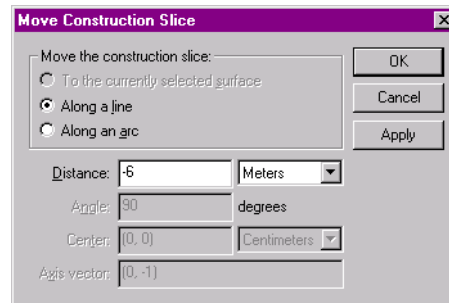
1. On the Edit menu, click Select Construction Slice Edges.
2. On the Edit menu, click Select All.
3. Press the Delete key.

## Move the construction slice

The air box begins below the starting surfaces of the C-core.

1. On the Draw toolbar, select the Move Construction Slice tool .


The Move Construction Slice dialog appears.

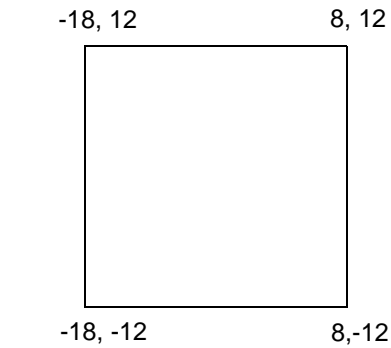


2. Make sure Along A Line is selected.
3. In the Distance box, enter -6.
4. Click OK.


## Draw the geometry

The grid should be enlarged before drawing the air box.

1. On the View menu, click Update Automatically.
2. On the Draw toolbar, select the Line drawing tool .
3. Using the mouse pointer, draw the geometry shown in the diagram below.



## 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 the Make Component in a Line tool .
4. In the Distance box, enter **13**.
5. In the Material drop down list, make sure **AIR** is selected.
6. In the Name box, enter **Air box**.
7. Click OK.
8. Click Save.

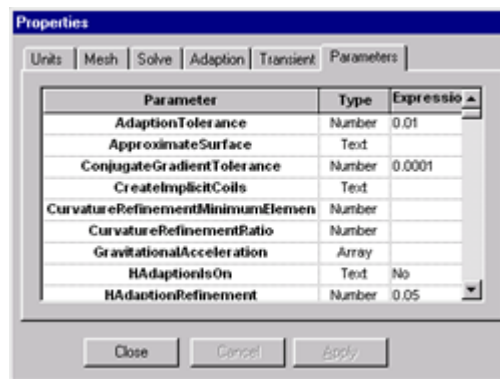
## Step 4: Parameterize the rotating block

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.

### Open the Model parameters page

1. In the Object page, click the name of the model (in this case, C-core with a rotating block.mn).
2. On the Edit menu, click Properties.  
The Properties dialog appears.
3. Click the Parameters tab.  
The Model parameters page appears.

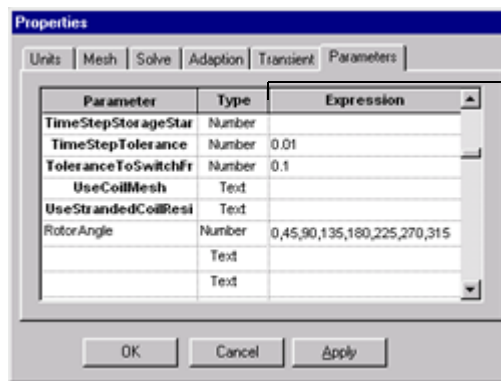


## Create a user-defined parameter

1. In the Parameters page, use the scroll bar to display an empty line.
2. In the Parameter column of the empty line, type **RotorAngle**.  
Make sure you do not add any spaces before, between, or after the words.

RotorAngle	Text	
------------	------	--

3. Click the mouse pointer in the Type column.  
A drop down list appears.
4. Choose Number from the drop down list.
5. In the Expression column, type the following: **0, 45, 90, 135, 180, 225, 270, 315**  
You can resize the columns by dragging the column separators.



click the mouse pointer  
here to drag the  
column separator

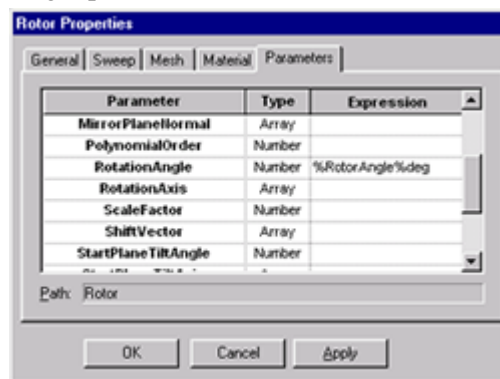
6. Click Apply.  
The next step is to use the RotorAngle parameter as an expression in the Component property page of the rotor.

## Apply the user-defined parameter

The position of the rotor is parameterized in the Component property page.

### Open the Component property page

1. On the Object page, select Rotor.
2. On the Edit menu, click Properties.  
The Properties dialog for the rotor is displayed.
3. Select the Parameters tab.
4. Use the scroll bar on the right side of the page to scroll down to the RotationAngle parameter.



5. In the Expression column of the Rotation Angle parameter, enter the following:

**%RotorAngle%deg**

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.

<b>RotationAngle</b>	Number	%RotorAngle%deg
----------------------	--------	-----------------

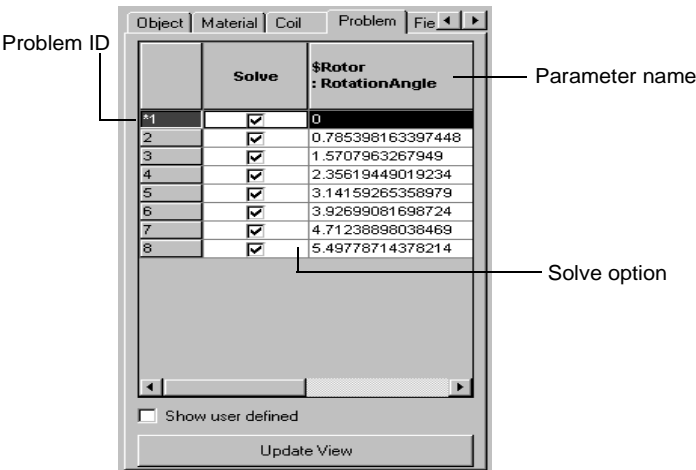
6. Press TAB.
7. Click OK.



## View the instantiated models


The instantiated models are viewed in the Problem page of the Project bar.

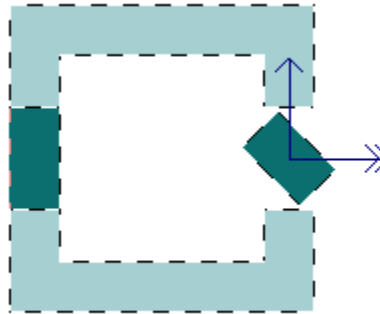
1. On the Project bar, click the Problem tab.  
The Problem page appears. (The angle is expressed in radians in the Problem page.)



2. Click the Problem 2 tab **2** in the second row in the Problem page.  
The row is highlighted.

*1	<input checked="" type="checkbox"/>	0
2	<input checked="" type="checkbox"/>	0.785398163397448
3	<input checked="" type="checkbox"/>	1.5707963267949

3. In the Problem page, click the Update View button . The View window is updated to display Problem 2.

**Tip**

The rotation angle is counterclockwise around the rotation axis.

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.
5. Click Save.

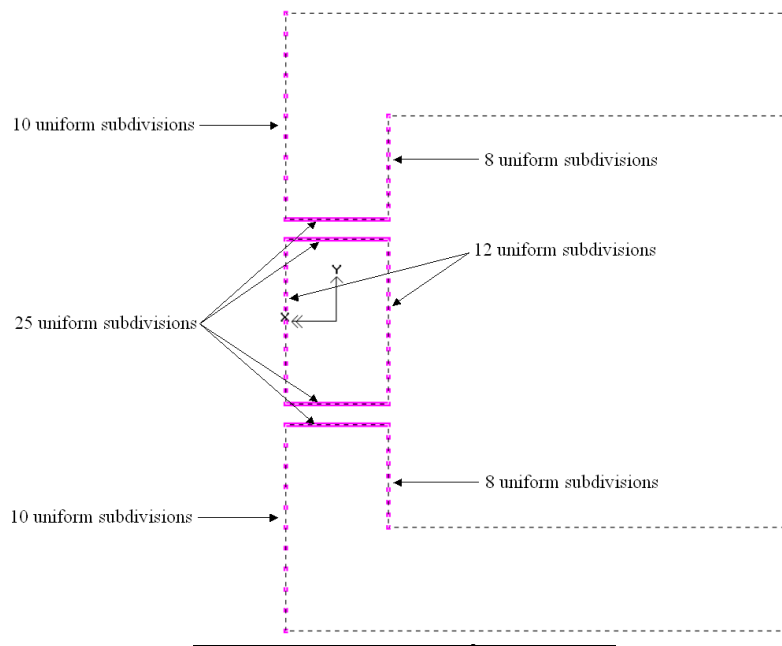
## Step 5: Modify the mesh

---

In the finite element method of analysis, the model is divided into a mesh of tetrahedral-shaped elements. The 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 subdivide component edges into segments. The number of edge segments corresponds to the number of elements along the edge. Note that the subdivisions propagate throughout the entire model in the sweep direction.

The edge subdivisions that are modified in this model are shown in the diagram below.




## Set the meshing method to extrusion

**Note:** If you do not have the Full 3D Solid Modeler license, this option is not available to you. Your default setting is extrusion.

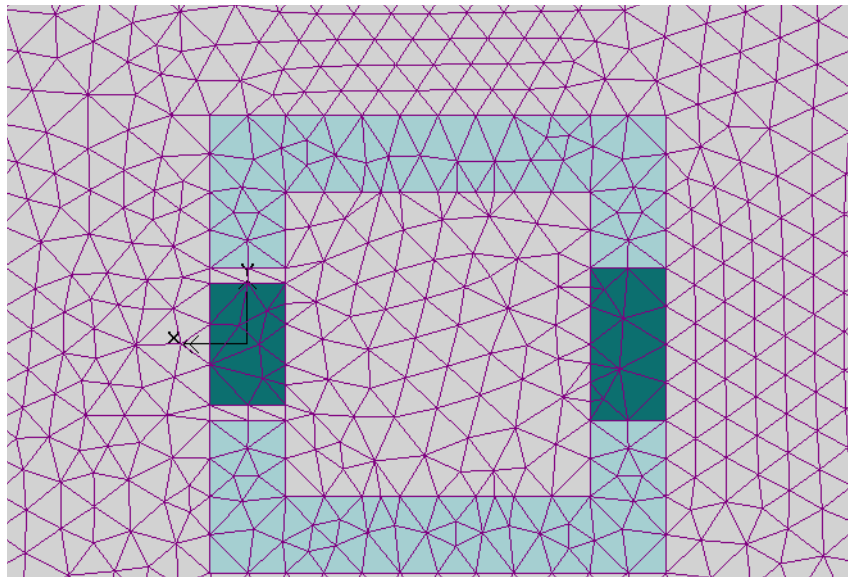
1. From the Object page, select **C-core with a rotating block**.
2. On the Edit menu, click Properties.
3. Select the Mesh page.
4. In the Meshing Method To Use drop down list, select Extrusion.
5. Click Ok.

## View the initial mesh

Before changing the subdivisions, the default initial mesh can be viewed.

1. On the Preset View toolbar, click  (negative Z axis).
2. On the View menu, click Initial 2D Mesh.

The initial mesh appears in the View window. The mesh should look like the following diagram.



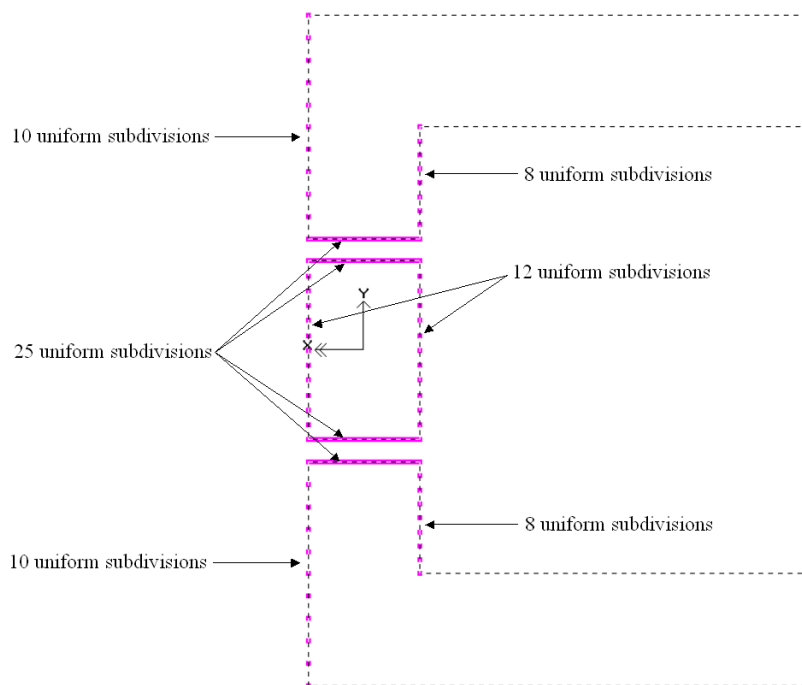
## Modify the edge subdivisions

Subdivisions can be applied to an edge using either a logarithmic or uniform (linear) scale. Note that subdivisions are edited on the model, not directly on the mesh. Subdivisions are applied to the model using the Mesh toolbar. Please see the Help for instructions on displaying toolbars.



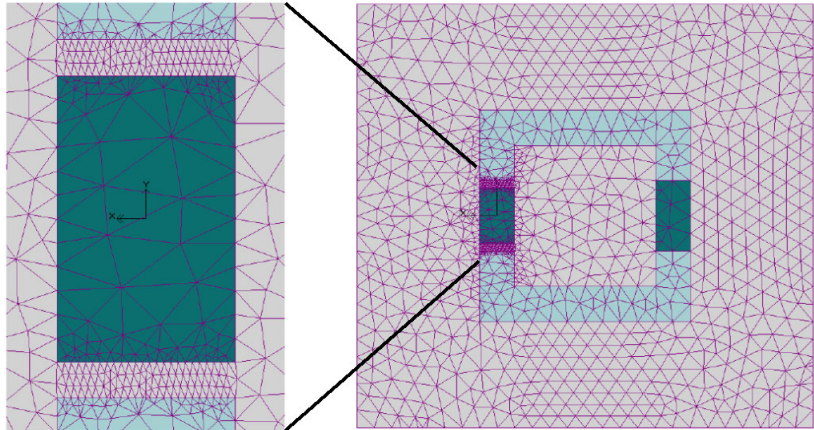
### Subdivide the air gap edges between the Rotor and the two Core components

1. On the View menu, click Solid Model.
2. On the Edit menu, click Select Component Edges.
3. Click the mouse pointer on the appropriate edges and assign the uniform subdivisions (as shown in the following diagram), using the Number of Subdivisions text box (the left text box)  , and then clicking the Uniform Edge Subdivision toolbar button.



**View the changes to the mesh**

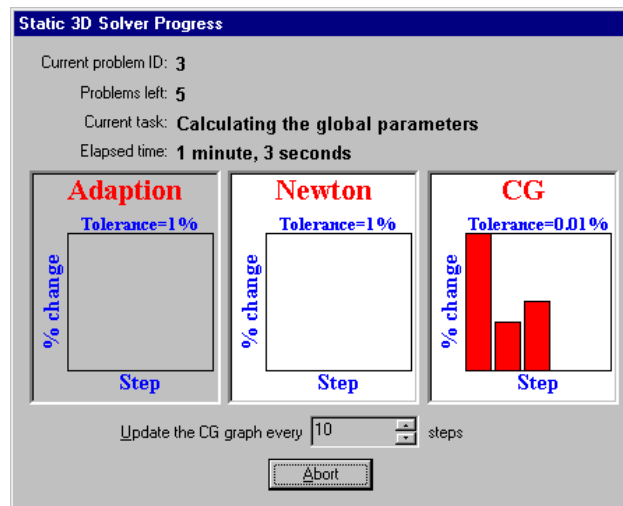
- On the View menu, click Initial 2D Mesh.  
The mesh updates (this may take a moment).  
The mesh should look like the following diagram.



## Step 6: Solve

---

- On the Solve menu, click Static 3D.  
The Static 3D Solver Progress dialog appears.



The model takes a few minutes to solve (solving time may vary according to computer). The Solver Progress dialog automatically exits when the solution is complete.

# Step 7: View the solution results

In this section, the force and torque on the rotating block is displayed in the Post Processing bar. 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.

## View the force on the rotating block

The Post Processing bar is automatically displayed at the bottom of the Main window when the solution is complete.

1.

Click the Force tab on the Post Processing bar.  
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.)

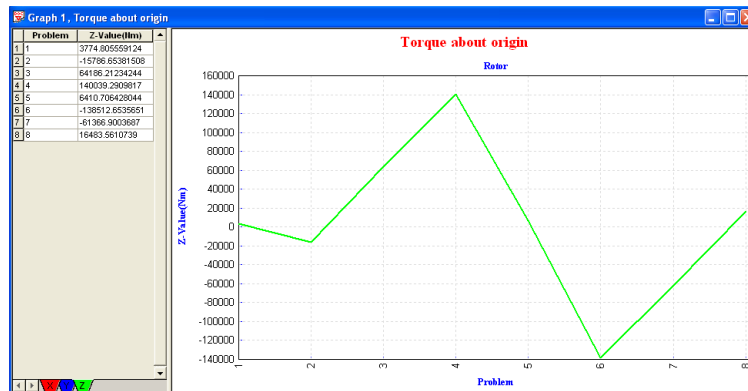
**Tip** Adjust the height of the Post Processing bar by placing the cursor on its top edge, then while keeping the left mouse button pressed, dragging the bar up or down.


Problem ID: Problem 3		Time:		Graph selection:	
Energy Force Flux Linkage Power Loss					
Force					
		Vector	Magnitude	Torque	
				Vector	Magnitude
Core upper half • Core lower half •		(2485.488419877, 1322.748542748, 241.2859777932)	2834.688902421	(-886.9815858559, 4140.634887206, -61634.85385973)	61781.83025893
Permanent Magnet					
Rotor		(-7949.01965189, 503.969234397, 559.3472178933)	7985.392087886	(-2425.900580789, -3864.027852452, 64186.21234244)	64340.15846669
Force Unit: Newtons		Torque Unit: NewtonMeters		Torque Center: (0, 0, 0)	
				Meters	



## Graph the Z-component of the torque

1. Click the mouse pointer in the torque vector entry for the Rotor.
2. Click the Graph Selection button .  
A Graph view appears. Each component of the vector is graphed separately.
3. Click the green tab  to display the Z component.  
The graph updates.



4. Close the Graph view by clicking on the Close icon  at the top right corner of the Graph view window.

## View the shaded plot on a slice through Problem 3

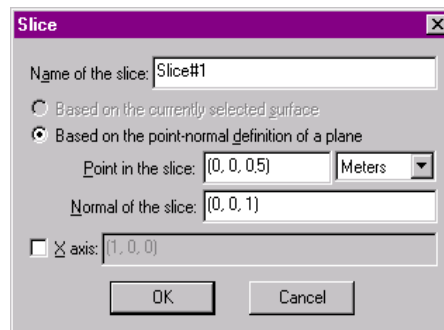
### Hide the display of the all components

The shaded plot is visible on the surface of all the components toggled visible in the Object page. In this case, the air box should be hidden from the display so that the C-core is more easily viewed.

1. In the Object page, select Core Upper Half, Core Lower Half, Permanent magnet, Rotor, and 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 red X is displayed next to all the components.  
All components are hidden from the display.

### Make a slice

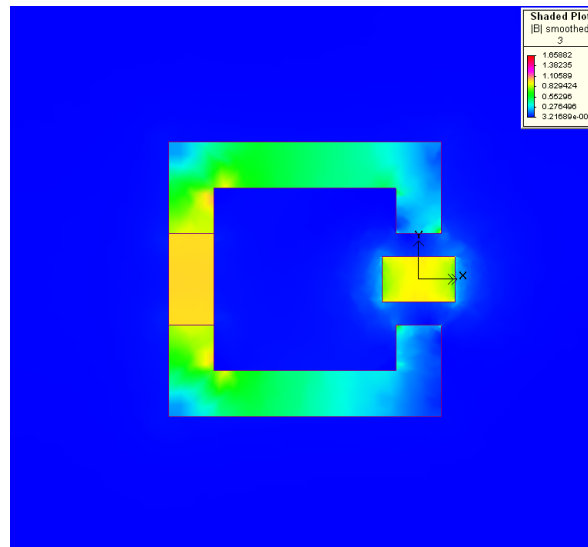
1. On the Tools menu, click New Slice.  
The Slice dialog box appears.



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.

### View the shaded plot

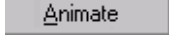
1. On the Project bar, select the Field tab.  
The Field page opens.
2. In the Problem ID text box, enter **3**.  
You can also use the drop down list box to select Problem 3.
3. Click Shaded Plot tab.
4. In the Fields To Display list, make sure that  $|\mathbf{B}|$  smoothed is selected.  
 $|\mathbf{B}|$  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.



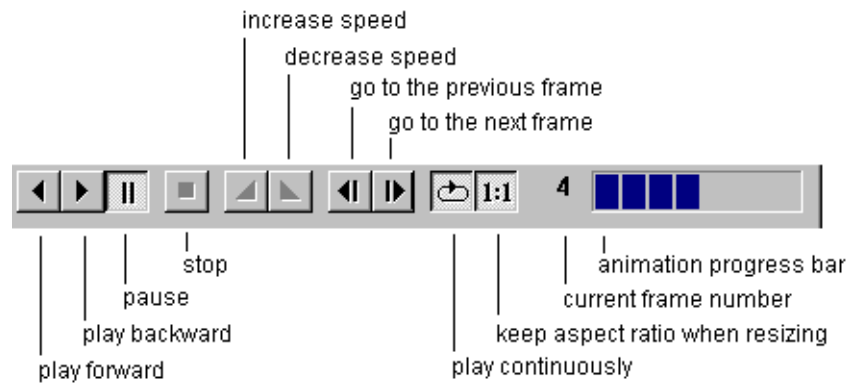
## Animate the shaded plot


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

**Tip** The color legend that is displayed during the animation represents the range of field values over the entire animation.

1. On the Field page, select the 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.

### 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.

### Save the model

You have now completed the tutorial.

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

## Summary

In this tutorial, you parameterized the position of a permanent magnet 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



# Index

- A
- Animate
  - Shaded plot 63, 216, 242
- Arcs
  - Draw 24, 76, 165, 182
- Arrow plot
  - View 154
- B
- Boundary condition
  - Bath plate boundary condition 168
  - C-core with a permanent magnet 227
  - Cylindrical Shield 15, 179
  - Pot-core with a coil 140
  - Spherical Shield 69
- C
- Calculator
  - Probe the magnitude of B at (0,0) 47, 101, 102, 113, 114
- Circuits
  - Add a coil 55, 206
  - Add a current source 54, 205
  - Add connections 55, 206
  - Circuit toolbars 53, 204
  - Edit an exponential waveform 57, 208
- Coils
  - Current-driven coils 163
  - Edit the properties of a coil 39, 96, 139, 201
  - Make Coil command 138
  - Voltage-driven coils 163
- Components
  - Change the material 52, 95, 108
- Contour plot
  - Animate 63, 216
  - View 99
- Current-driven coils
  - Create 163
- D
- Distortion 124, 125, 126
- E
- Edge subdivisions
  - Modify 30, 83, 145, 192, 237
- Energy page
  - Pot-core with a coil 152
- Exponential waveform
  - Define the start, stop and step times 58, 209
  - Diagram 57, 208
  - Edit 59, 210
- F
- Field page 44, 62, 99, 111, 215
- Field values
  - Probe 47, 101, 102, 113, 114
- Finite element method of analysis
  - 2D analysis 28, 81, 190, 235
  - 3D analysis 143, 190, 235
- Flux function
  - View 63, 215
- Force page

- C-core with a rotating block 240
- G
- Graph
  - Power across time 65, 218
  - Z-component of the torque 241
- I
- Initial mesh
  - View 29, 82, 144, 191, 236
- M
- Materials
  - Change the material of a component 52, 95, 108
  - Create a new material 107, 161
- Mesh
  - Modify mesh edge subdivisions 30, 83, 145, 192, 237
  - View the initial mesh 29, 82, 144, 191, 236
- Model units
  - Set the model units 118
- P
- Parameter page
  - Re-size 231
- Parameterization
  - User-defined parameters 230
- Permanent magnet
  - Set the direction of magnetization 224
- Polynomial order
  - Change the polynomial order of the model 42, 203
- Power
  - Graph across time 65, 218
  - View the instantaneous power 64, 217
  - View the time-averaged Ohmic power 44, 62, 112, 173, 215
- Problem page
  - View 233
- R
- Re-size
  - Field page 62, 215
  - Parameter page 231
- S
- Set Solver Options dialog 41, 109, 202
- Set the model units 159
- Set Transient Options dialog 58, 209
- Shaded plot
  - Animate 63, 216, 242
- Slice
  - Make a slice based on a surface of the model 154
  - Make a slice based on the point normal definition of a plane 242
- Smooth a field
  - Smooth the magnitude of Bz 175
- Solve
  - Magnetostatic 2D 98
  - Magnetostatic 3D 151
  - Time-harmonic 2D 43, 61, 110, 213, 239
  - Time-harmonic 3D 172
  - Transient 2D 43, 61, 98, 110, 213, 239
- Source frequency



- Set the source frequency 41, 109, 171, 202
- Symmetry conditions
  - Cylindrical shield 67
  - Pot-core with a coil 117
  - Spherical shield 68
- T
- Time unit
  - Change the default time unit 63, 215
- U
- Units
  - Change the default time unit 63, 215
  - Set the model units 118
- User-defined parameters 230
  - Create 231
- V
- Voltage-driven coils
  - Create 163
  - Edit properties 167
- W
- Waveform
  - Open the waveform editor 210