Version 6.11.1

2D Transient with Motion
Team Problem 30
Tutorial

Infolytica Corporation

Octoner 8, 2002

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
P.O. Box 1144
Station Place du Parc
Montreal, Quebec H2X 4B3
Canada

Part number TUT_TR2D_Motion_004.

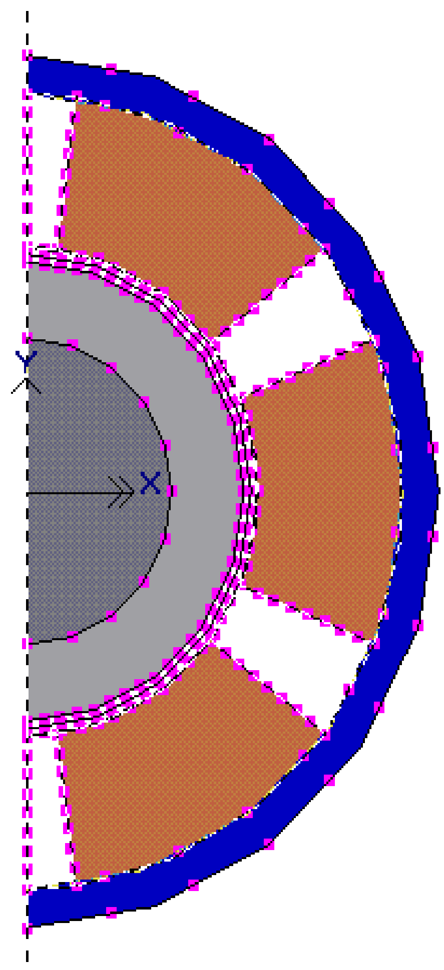
Table of Contents

2D Transient with Motion Tutorial: Team Problem 30

Modeling plan

The devices described in the TEAM problem 30[1] are appropriate applications of the transient solver. One of the devices described in the TEAM problem 30 is a 3-phase induction machine. Most of the analyses of this device can be done using the time harmonic solver. While using the time harmonic solver is the most economical and preferred method, there are conditions that require the use of a transient solver. The intention of this tutorial is to examine one or more of these conditions and demonstrate what the steps are for setting up, solving and post-processing the solutions when a transient solver is used.

The following steps guide you through creating a half-model of the 3-phase induction machine and then analyzing the device in MagNet.



Step 1: Creating a new model

Opening a new model

- From your desktop, double-click the MagNet icon.

  The Main window appears.

  or

- If MagNet is already running, on the File menu, click New to open a new model.

Name the model

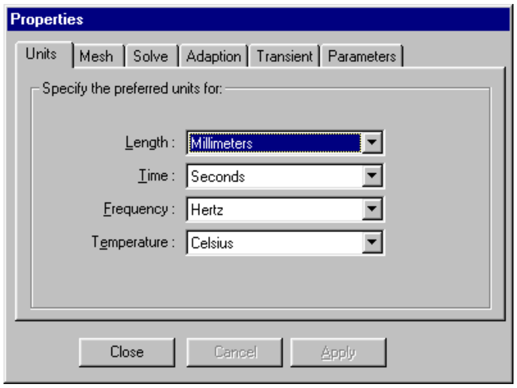
By default, MagNet assigns a name to the model (e.g., MagNet1) every time a new model is opened. As long as the application remains open, each new model number increments by one (e.g., if the new model you have opened is the fourth one in this session, MagNet would assign the name MagNet4). You can choose to retain this name, although it is recommended that you give the model a distinct name.

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

Step 2: Setting up the working environment

Initial Settings

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



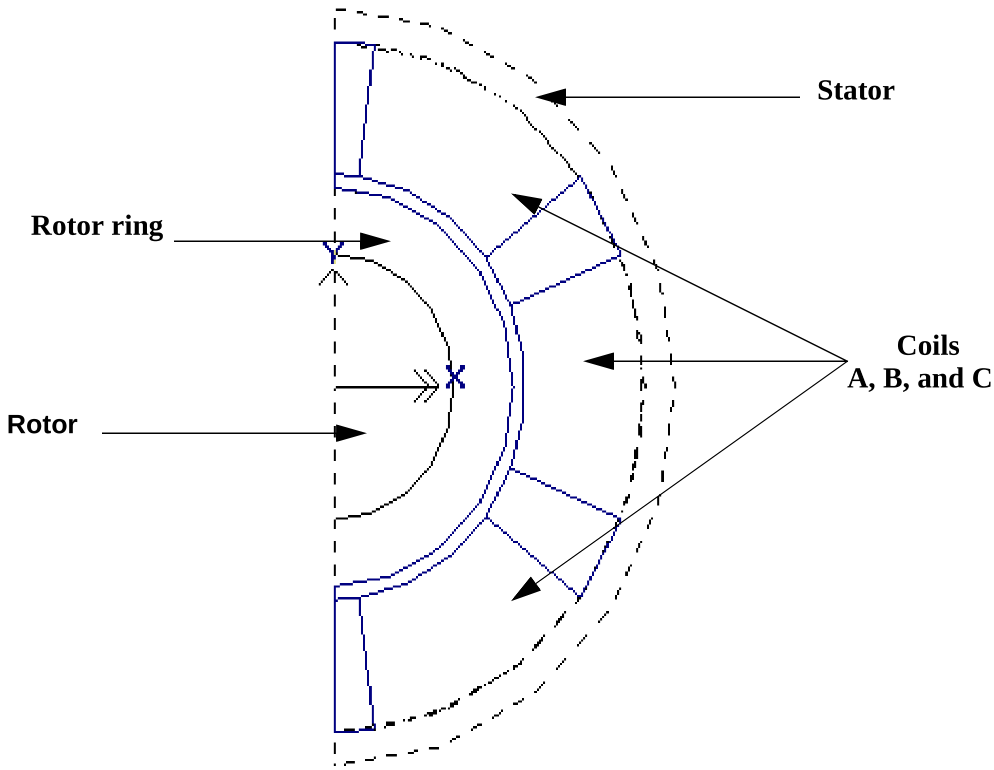
1. From the Object page, select the model (i.e., Team Problem 30).
2. On the Edit menu, click Properties.
3. From the Length drop down list, select **Centimeters**.
4. From the Time drop down list, select **Milliseconds**.
5. Click OK.

Set view to update automatically

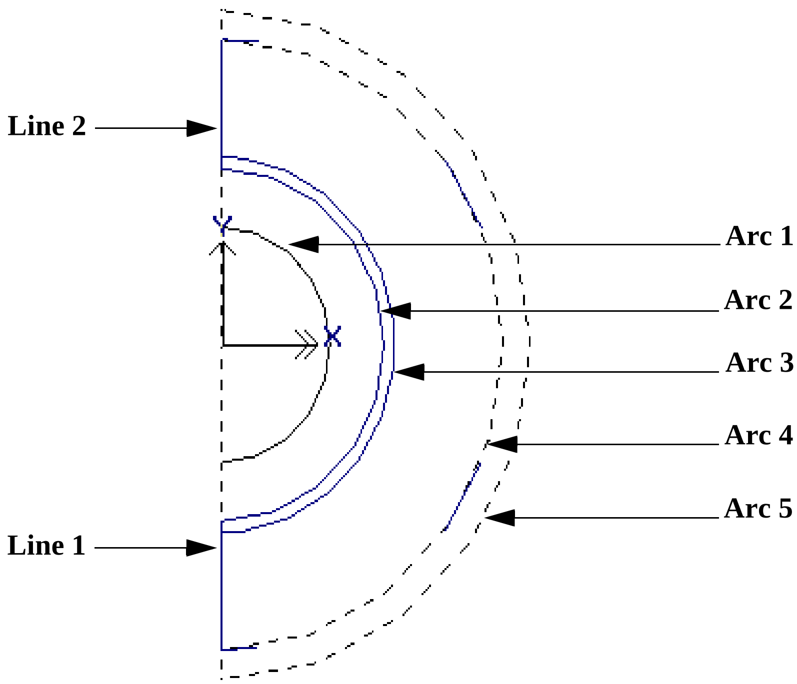
Use of this feature resets the view automatically to include all of the model's geometry, as it is being drawn.

- On the View menu, click Update Automatically.

Step 3: Build the geometric model and set up the problem for the Rotor, Rotor ring, Stator and the Coils



Draw the geometric model of the rotor, rotor ring, and the stator components



1. On the Tools menu, click Keyboard Input Bar.
2. On the Draw menu, click Line.
3. In the Keyboard Input bar, enter the following coordinates to draw the Lines 1 and 2:

Line 1

Start coordinates (**Cartesian**)	**0, -5.7**	Press ENTER
End coordinates (**Cartesian**)	**0, 0**	Press ENTER, and then ESC

Line 2

Start coordinates (**Cartesian**)	**0, 0**	Press ENTER
End coordinates (**Cartesian**)	**0, 5.7**	Press ENTER, and then ESC

4. On the Draw menu, click Arc.

5. In the Keyboard Input bar, enter the following coordinates to draw Arcs 1through 5.

<u>Arc 1</u>

Center coordinates (**Cartesian**)	**0, 0**	Press ENTER
Start coordinates (**Cartesian**)	**0, -2.0**	Press ENTER
End coordinates (**Cartesian**)	**0, 2.0**	Press ENTER

<u>Arc 2</u>

Center coordinates (**Cartesian**)	**0, 0**	Press ENTER
Start coordinates (**Cartesian**)	**0, -3.0**	Press ENTER
End coordinates (**Cartesian**)	**0, 3.0**	Press ENTER

<u>Arc 3</u>

Center coordinates (**Cartesian**)	**0, 0**	Press ENTER
Start coordinates (**Cartesian**)	**0, -3.2**	Press ENTER
End coordinates (**Cartesian**)	**0, 3.2**	Press ENTER

<u>Arc 4</u>

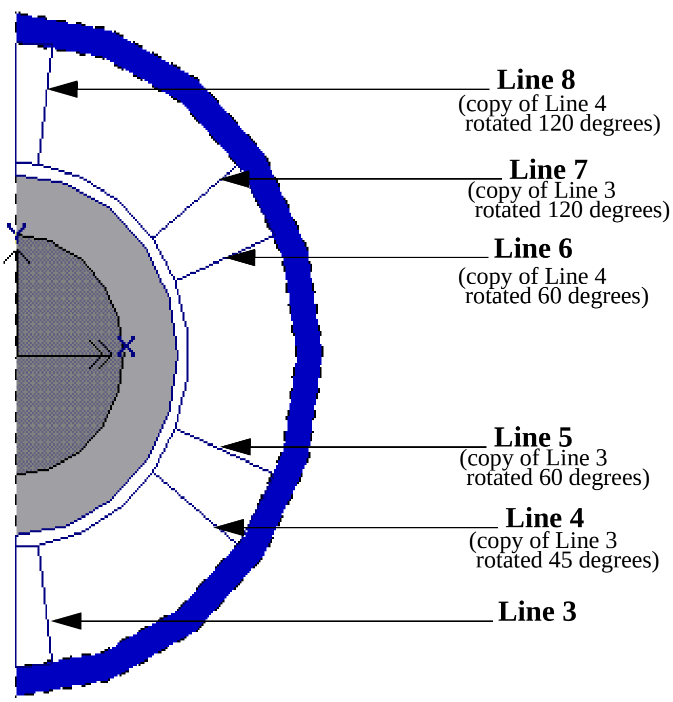
Center coordinates (**Cartesian**)	**0, 0**	Press ENTER
Start coordinates (**Cartesian**)	**0, -5.2**	Press ENTER
End coordinates (**Cartesian**)	**0, 5.2**	Press ENTER

Arc 5

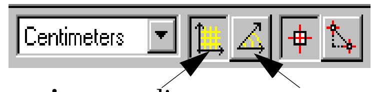
Center coordinates (**Cartesian**)	**0, 0**	Press ENTER
Start coordinates (**Cartesian**)	**0, -5.7**	Press ENTER
End coordinates (**Cartesian**)	**0, 5.7**	Press ENTER

Draw the geometric model of the coils

In this procedure, only one line (Line 3) is actually drawn. Using the Edge Transformation feature, this line is then duplicated to create lines 4, 5, and 7. Lines 6 and 8 are created using a copy of Line 4.



1. On the Draw menu, click Line.
2. In the Keyboard Input bar, enter the following polar coordinates to draw Line 3:



Click here for **cartesian** coordinates Click here for **polar** coordinates

<u>Line 3</u>

Start coordinates (**Polar**)	**3.2, -82.5**	Press ENTER
End coordinates (**Polar**)	**5.2, -82.5**	Press ENTER, and then ESC

3. On the Edit menu, click Select Construction Slice Edges.
4. Using the mouse pointer, select the line you have just drawn.
5. On the Draw menu, click Rotate Edges.
6. In the Rotation Angle text box, type **45.0**, and make sure that the following option is selected:
   - **Apply the transformation to a copy of the selection**
7. Click OK.
8. On the Draw menu, click Rotate Edges.
9. In the Rotation Angle text box, type **60.0**, and make sure that the following options are selected:
   - **Apply the transformation to a copy of the selection**
   - **Accumulate the Rotation angle on Apply**
10. Click Apply twice to create Lines 5 and 7, as shown above.
    Two copies of the line are created.
11. Click Cancel.
    Note In this case, if you were to click OK to close the Rotation Edges dialog, another copy of the line would be created (i.e., another line rotated 180 degrees from Line 3). Alternatively, to build the lines above, you could have clicked Apply once and then clicked OK to create the 2nd copy.
12. Using the mouse pointer, select Line 4 (shown in the illustration above).
13. On the Draw menu, click Rotate Edges.
14. In the Rotation Angle text box, type **60.0**, and make sure that the following options are selected:
    - **Apply the transformation to a copy of the selection**
    - **Accumulate the Rotation angle on Apply**
15. Click Apply twice to create Lines 6 and 8, as shown above.
    Two copies of the line are created.
16. Click Cancel.

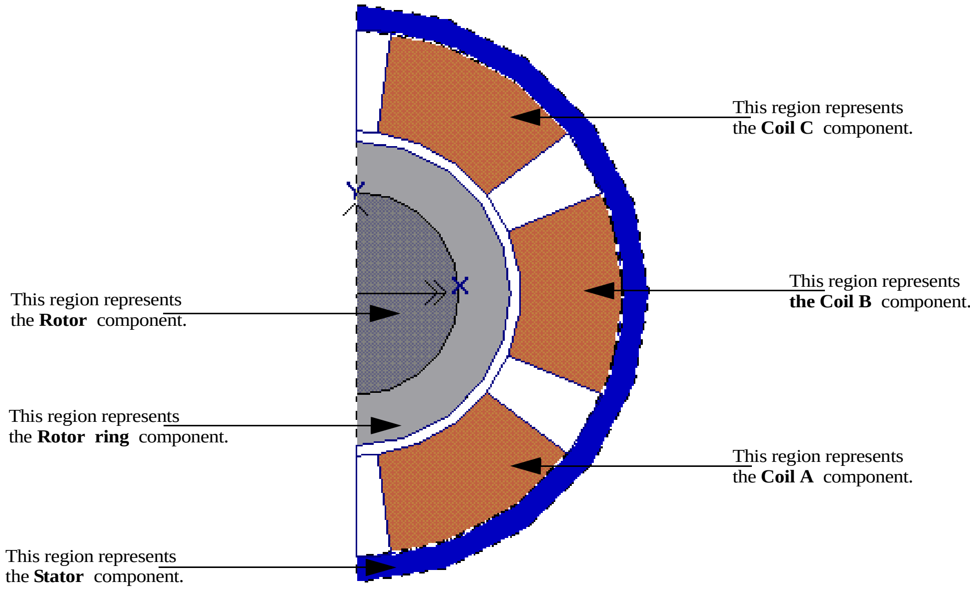
Create new materials

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

1. On the Tools menu, click New User Material.
2. On the General page, enter the following data:
   - Name: **Rotor Steel**
   - 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***
   - Other ***Mass Density***
5. Using the Next button to advance to the appropriate pages, enter the following values:
   - Temperature *Celsius* = **20**
   - Relative Permeability = **30**
   - Coercivity *Amps/m* = **0**
   - Conductivity *Siemens/m* = **1.6e6**
   - Mass Density *Kg/m^3* = **7800**
6. Once you have entered all the values, click Finish in the Confirmation page to create the new material.
7. Repeat steps 1 through 6 using the data listed below:

• Name: **Rotor Aluminum**	• Name: **Stator Steel**
• Display color: *Click Set Color and select an appropriate color*	• Display color: *Click Set Color and select an appropriate color*
• Temperature *Celsisus* = **20**	• Temperature *Celsisus* = **20**
• Set Relative Permeability to **1**	• Set Relative Permeability to **30**
• Coercivity *Amps/m* = **0**	• Coercivity *Amps/m* = **0**
• Set Conductivity to **3.72e7** *Siemens/m*	• Set Conductivity to **0** *Siemens/m*
• Mass density: **2707** *Kg/m^3*	• Mass density: **7800** *Kg/m^3*

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



1. On the Edit menu, click Select Construction Slice Surfaces.
2. Select the interior region for **Rotor** (as indicated above) of the model you have drawn.
3. On the Model menu, click Make Component in a Line, and enter the following values:
   - Distance **100** Centimeters
   - Material: **Rotor Steel**
   - Name: **Rotor**
   - Make sure that the following option is **not selected**, since the vertex at the origin is needed when we will assign periodic boundary conditions later on in this document:

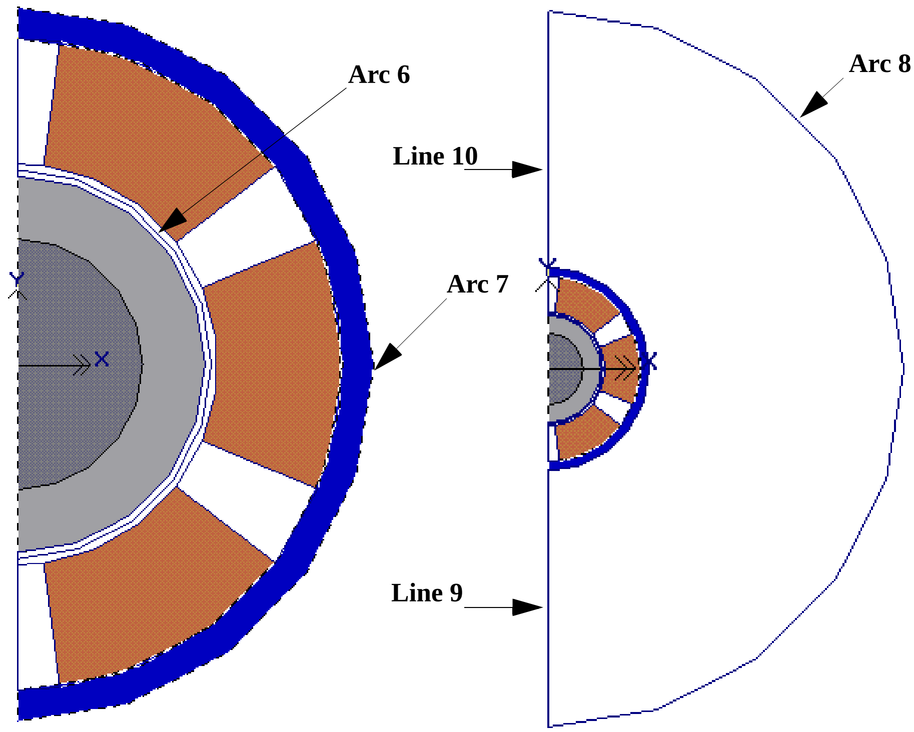
     Remove unnecessary vertices on the boundaries
4. Click OK.
5. Select the interior region for **Rotor ring** (as indicated above) of the model you have drawn.
6. On the Model menu, click Make Component in a Line, and enter the following values:
   - Distance **100** Centimeters
   - Material: **Rotor Aluminum**
   - Name: **Rotor ring**

   Note Reset the option **Remove unnecessary vertices on the boundaries** and leave it selected for all other components.
7. Click OK.

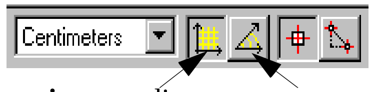
8. Select the interior region for **Stator** (as indicated above) of the model you have drawn.
9. On the Model menu, click Make Component in a Line, and enter the following values:
   - Distance **100** Centimeters
   - Material: **Stator Steel**
   - Name: **Stator**
10. Click OK.
11. Select the interior region for **Coil A** (as indicated above) of the model you have drawn.
12. On the Model menu, click Make Component in a Line, and enter the following values:
    - Distance **100** Centimeters
    - Material: **Copper: 5.77e7 Siemens/meter**
    - Name: **Coil A**
13. Click OK.
14. Repeat steps 11 through 13 to make **Coil B** and **Coil C**, applying the same distance and material.
15. On the File menu, click Save.

Draw the geometric model for the volume of air

For this type of problem, where the periodic boundary condition is used, it is necessary to divide the air gap between the Stator and the Rotor ring components. An air box to surround the model is also required since the device has an unusually low permeability for the stator. Given that the Stator is very thin, there will likely be some field leaking outside the device that, if not modeled, will affect the results.



1. On the Draw menu, click Line.
2. In the Keyboard Input bar, enter the following coordinates to draw the Lines 9 and 10:



Click here for **cartesian** coordinates Click here for **polar** coordinates

<u>Line 9</u>

Start coordinates (**Cartesian**)	**0, -5.7**	Press ENTER
End coordinates (**Cartesian**)	**0, -20.0**	Press ENTER, and then ESC

<u>Line 10</u>

Start coordinates (**Cartesian**)	**0, 5.7**	Press ENTER
End coordinates (**Cartesian**)	**0, 20.0**	Press ENTER, and then ESC

3. On the Draw menu, click Arc.
4. In the Keyboard Input bar, enter the following coordinates to draw Arcs 6, 7 and 8.

<u>Arc 6</u>

Center coordinates (**Cartesian**)	**0, 0**	Press ENTER
Start coordinates (**Cartesian**)	**0, -3.1**	Press ENTER
End coordinates (**Cartesian**)	**0, 3.1**	Press ENTER

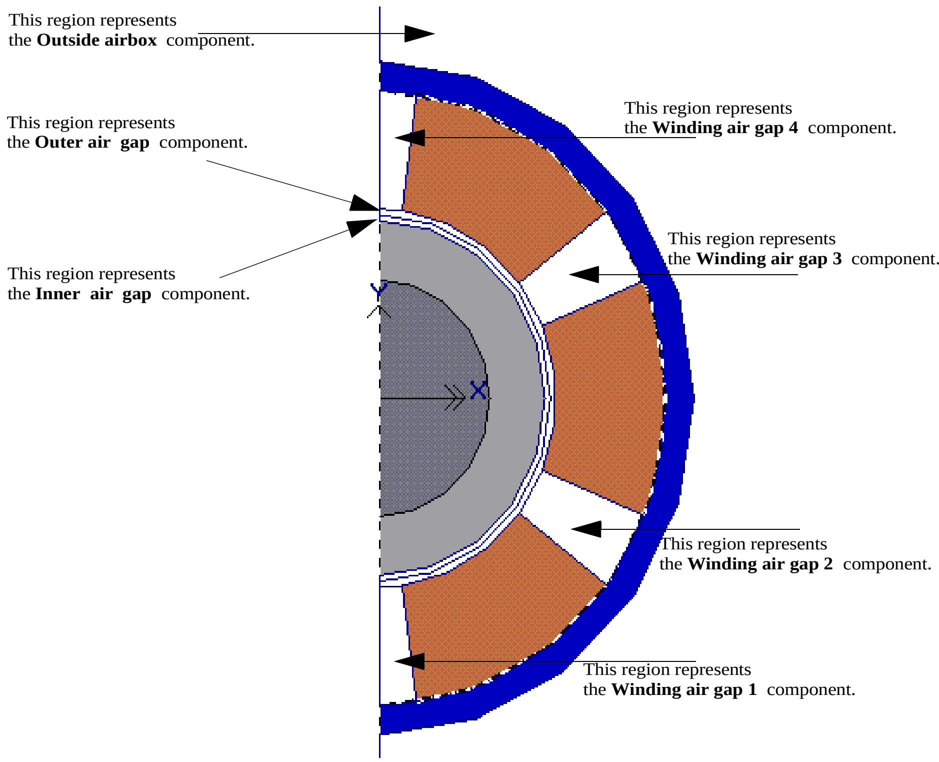
<u>Arc 7</u>

Center coordinates (**Cartesian**)	**0, 0**	Press ENTER
Start coordinates (**Cartesian**)	**0, -5.7**	Press ENTER
End coordinates (**Cartesian**)	**0, 5.7**	Press ENTER

Arc 8

Center coordinates (**Cartesian**)	**0, 0**	Press ENTER
Start coordinates (**Cartesian**)	**0, -20.0**	Press ENTER
End coordinates (**Cartesian**)	**0, 20.0**	Press ENTER

Make the air components



1. On the View menu, click Zoom, and then click Zoom In At Point/Box.
2. Keeping the mouse button clicked, drag the cursor to form a rectangle around the regions labelled for the Inner and Outer air gap (as shown above on the illustration).
3. On the Edit menu, click Select Construction Slice Surfaces.
4. Select the interior region for **Inner air gap** (as indicated above) of the model you have drawn.
5. On the Model menu, click Make Component in a Line, and enter the following values:
   - Distance **100** Centimeters
   - Material: **AIR**
   - Name: **Inner air gap**
6. Click OK.

7. Repeat steps 4 through 6 to make **Outer air gap,** applying the same distance and material.
8. On the View menu, click Update Automatically.
9. On the View menu, click Zoom, and then click Zoom In At Point/Box.
10. Keeping the mouse button clicked, drag the cursor to form a rectangle around the model, excluding most of the Outside airbox.

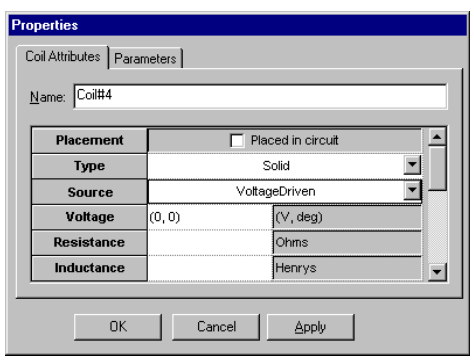
    The regions of the model that you see should be similar to what is shown in the illustration on the previous page.
11. Repeat steps 3 through 6 to make **Winding air gap 1 to 4** and **Outside airbox**, applying the same distance and material.

Make coils

1. From the Object page, select the Coil A component.
2. On the Model menu, click Make Simple Coil.

   Coil#1 is created and listed in the Object page.
3. Repeat steps 1 and 2 to create coils from the following components:
   - **Coil B** is selected to create **Coil#2**
   - **Coil C** is selected to create **Coil#3**
   - **Rotor** is selected to create **Coil#4**
   - **Rotor ring** is selected to create **Coil#5**

Note The reason for making coils out of the **Rotor** and **Rotor ring** components is that only half of the problem is modeled and the system needs to know that these two components are electrically shorted at the end. This would not be necessary if you were modeling the entire device.



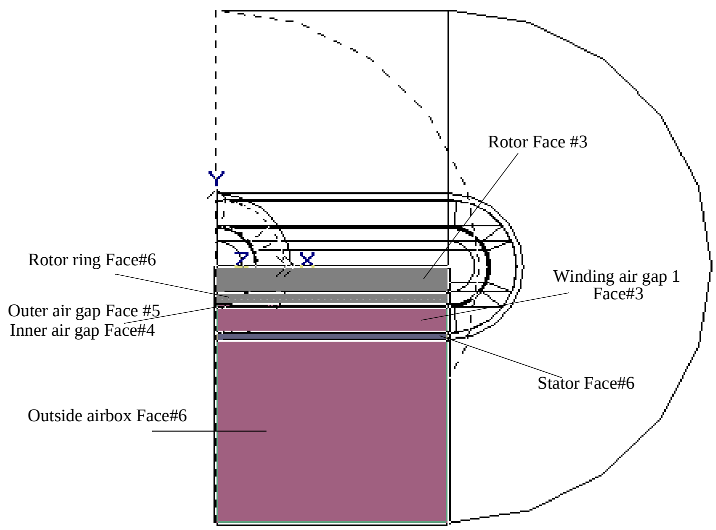
4. From the Object page, select **Coil#4**.
5. On the Edit menu, click Properties.

   The Coil Properties dialog appears.
6. In the Coil Attributes page, do the following:
   - for Type, select **Solid**
   - for Source, select **VoltageDriven**

7. Click Apply.
8. From the Object page, select **Coil#5**.
9. In the Coil Attributes page, do the following:
   - for Type, select **Solid**
   - for Source, select **VoltageDriven**
10. Click OK.

Assign boundary conditions

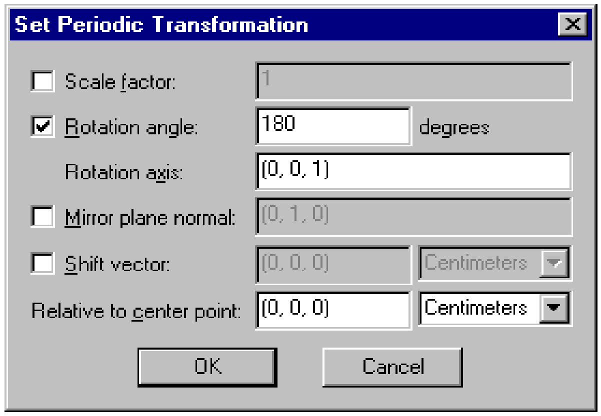
Since we have created only a half model of our device, we need to assign periodic boundary conditions. This allows the mesh nodes on one plane of symmetry to match the mesh nodes on the other side (the half of the device that is not modeled).



1. On the View menu, click Wireframe Model.
2. On the View menu, click Update Automatically.
3. On the View menu, click Rotate, and then Rotate Right.
4. On the Edit menu, click Select Component Surfaces.
5. From the Object page, keeping the CTRL key pressed, use the mouse pointer to click and select the following surfaces:
   - Rotor Face#5
   - Rotor ring Face#6
   - Stator Face#6
   - Inner air gap Face#6
   - Outer air gap Face#6
   - Winding air gap 1 Face#6
   - Outside airbox Face#6
6. On the Boundary menu, click Odd Periodic.

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

7. Click Set Transformation.

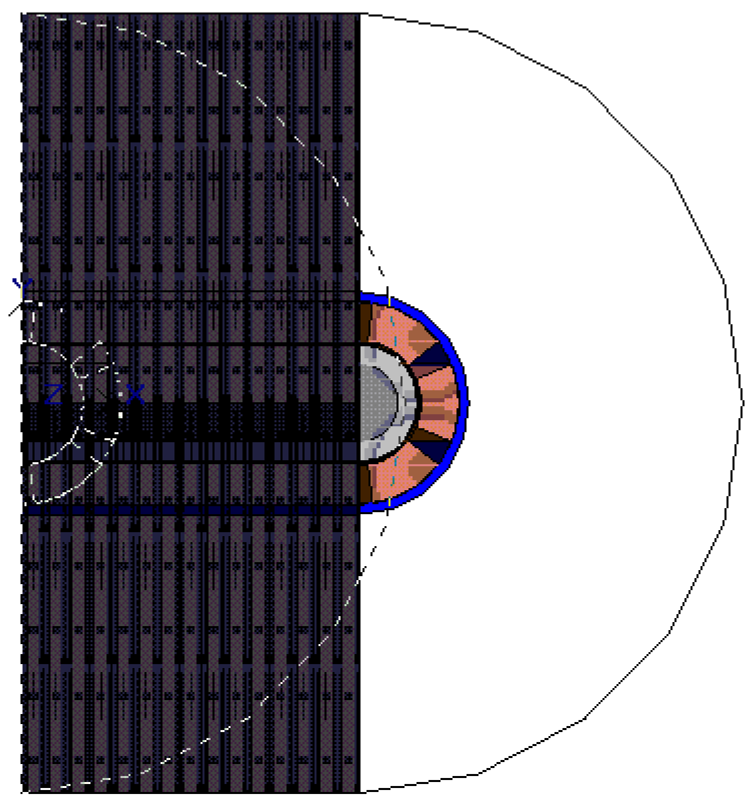


8. In the Set Periodic Transformation dialog, click inside the Rotation angle check box, and then type **180** in the degrees text box.

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

10. On the View menu, click Solid Model.

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

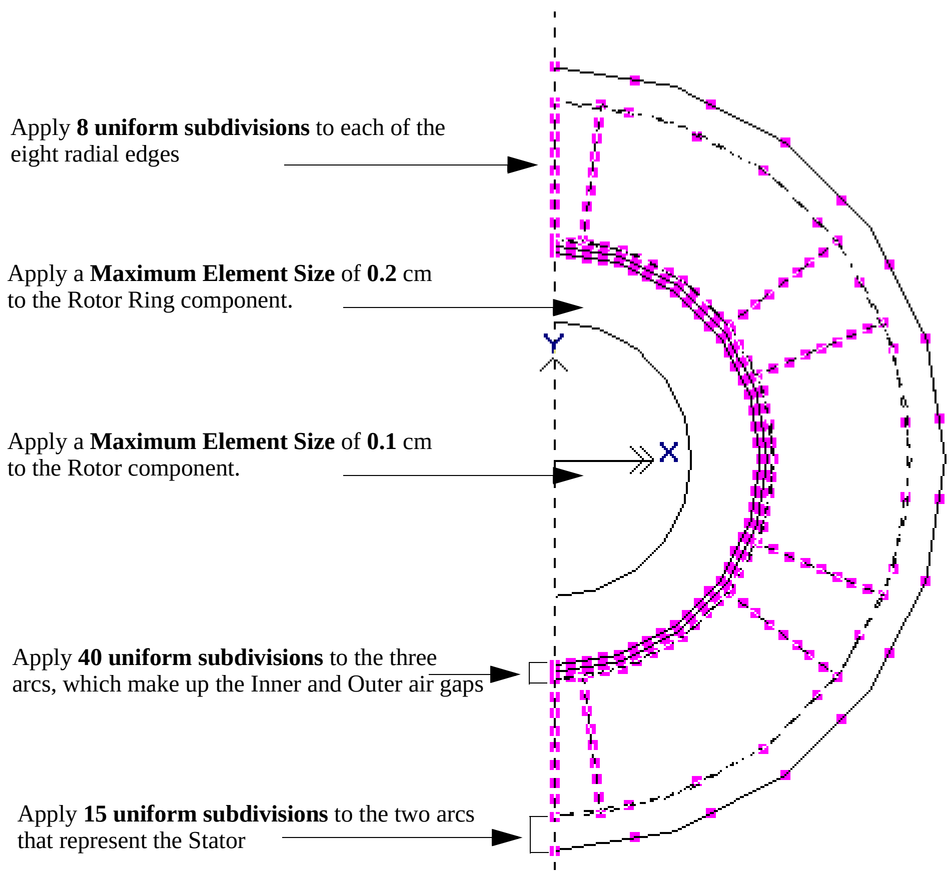


Modify the mesh

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 set the *maximum element size* for a specific component. Another method is to subdivide component edges into segments -- the number of edge segments correspond to the number of elements along the edge. The following procedures will demonstrate these two methods.

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

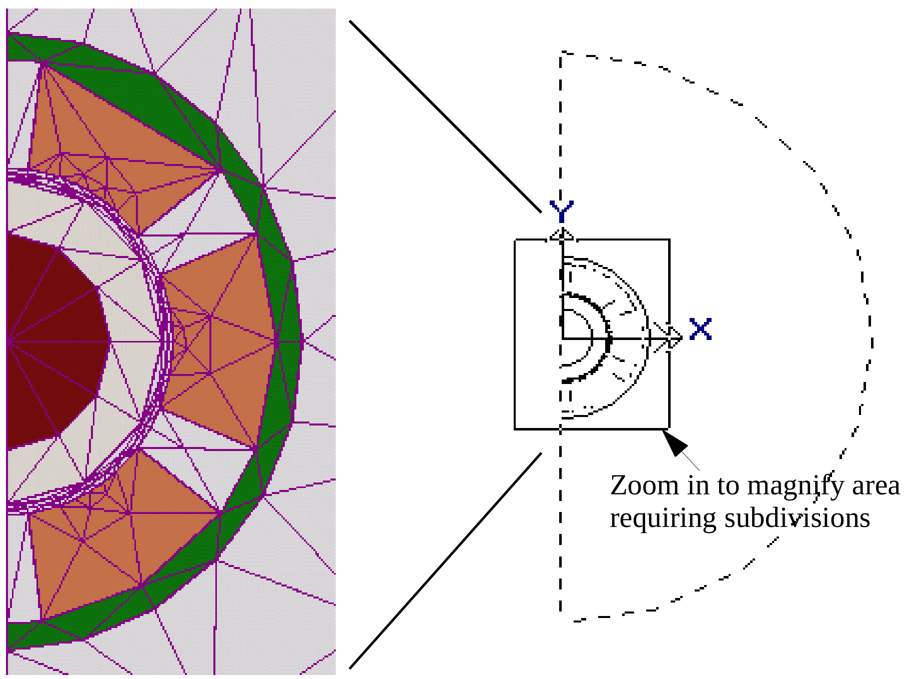


Note Before changing the *maximum element size* and the subdivisions, the default initial mesh can be viewed.

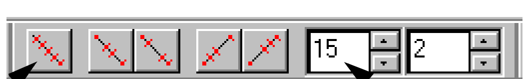
1. On the View menu, click Preset Views, and then Positive Z Axis.
2. On the View menu, click Zoom, and then click Zoom In At Point/Box.
3. Keeping the mouse button clicked, drag the cursor to form a rectangle around the model (as shown below on the illustration to the right), excluding the Outside airbox.
4. 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.

   The mesh should look like the illustration shown below on the left side.



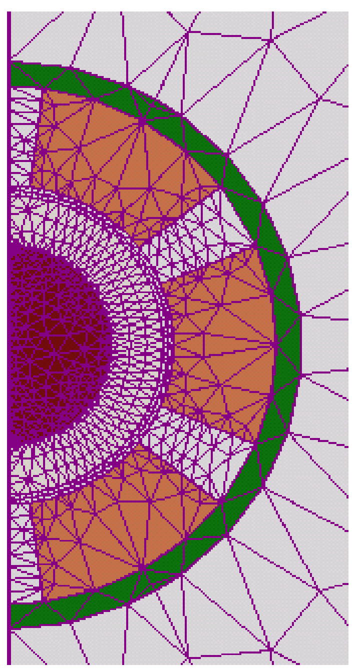
5. On the View menu, click Wireframe Model.
6. On the Edit menu, click Select Component Edges.
7. Using the illustration on page 18 as a guide, on the Mesh Toolbar enter the appropriate values (from the table on the next page) in the Number of Subdivisions text box, and then click the Uniform Edge Subdivision toolbar button.



Edge	Subdivisions
Keeping the CTRL key pressed, click the two arcs that represent the Stator	**Apply 15 uniform subdivisions**
Keeping the CTRL key pressed, click the eight radial edges	**Apply 8 uniform subdivisions**
Keeping the CTRL key pressed, click the three arcs that make up the Inner and Outer air gaps	**Apply 40 uniform subdivisions**

8. Once the subdivisions have been applied, select the **Rotor** component in the Object page.
9. On the Edit menu, click Properties, and then select the Mesh tab.
10. In the "Specify meshing control options" box, click inside the *Maximum element size* checkbox, and then type **0.1** in the text box.
11. Click Apply.
12. Select the **Rotor Ring** component in the Object page.
13. Click inside the *Maximum element size* checkbox, and then type **0.2** in the text box.
14. Click OK.
15. On the View menu, click Initial 2D Mesh.

    The initial mesh (with the new subdivisions) appears in the View window and should look like the illustration shown below.



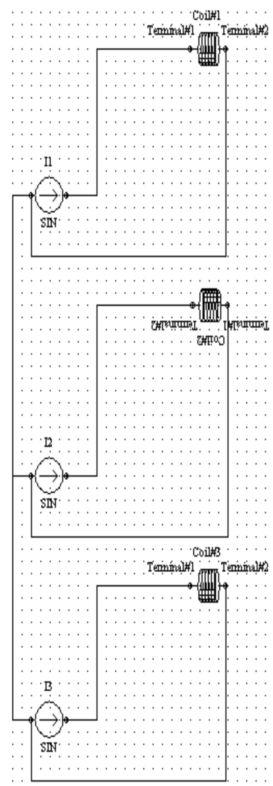
16. On the View menu, click Update Automatically.

Step 4: Setting up the problem for locked rotor analysis

The locked rotor analysis can be done using the time harmonic solver. Under this condition the rotor is locked in position when the torque is measured.

Creating the circuit for locked rotor analysis

Before doing this analysis, the driving circuitry should be defined. This is accomplished by creating the following 3 circuits.



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

   The Circuit window opens with Coil#1 through Coil#5 appearing in the left frame.
2. On the Circuit menu, click Zoom Percent, and then 75%.

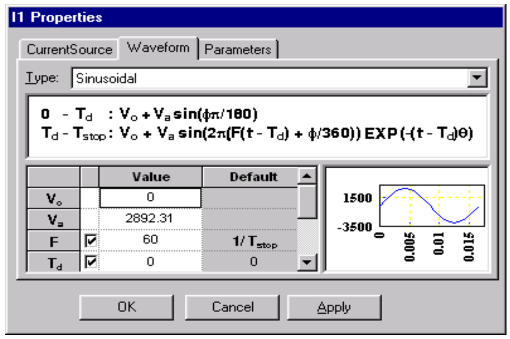
   This allows you to view the whole circuit area.
3. From the Edit menu, click Select.
4. Place the cursor over Coil#1, and keeping the mouse button clicked, drag the coil to the Circuit window.

   Repeat this step for Coil#2 and Coil#3, placing them as shown in the illustration above.
5. Using the mouse pointer, click on Coil#2 in the Circuit window.

6. On the View menu, click Rotate, and then Rotate Clockwise. Repeat this step once more so that Coil#2 is rotated 180 degrees.
7. On the Circuit menu, click Current Source.
8. Position the cursor in the Circuit window, and then click once at each of these locations (down and to the left of Coil#1, Coil#2, and Coil#3) to create three current source components (I1, I2, and I3).
9. On the Circuit menu, click Connection, and then do the following:

Click	then click
I1 - Right Terminal	Coil#1 - T1
Coil#1 - T2	I1 - Left Terminal
I2 - Right Terminal	Coil#2- T2
Coil#2 - T1	I2 - Left Terminal
I3 - Right Terminal	Coil#3 - T1
Coil#3 - T2	I3 - Left Terminal
I1 - Left Terminal	I2 - Left Terminal
I2 - Left Terminal	I3 - Left Terminal

Modifying the current source components

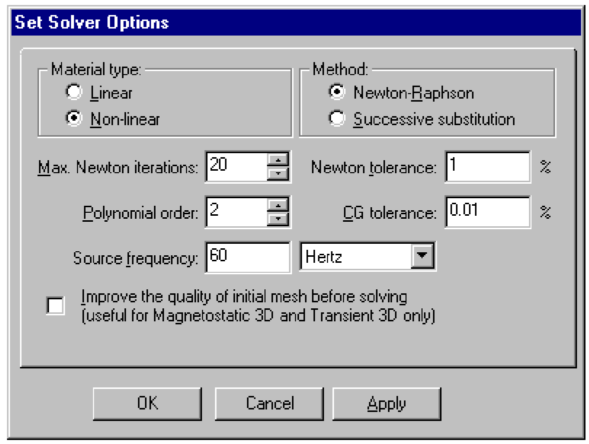


1. In the Object page, select **I1**.
2. On the Edit menu, click Properties.

   The Current Source Properties dialog appears.
3. Open the Waveform page and make the following modifications:
   - In the V_a text box, type **2892.31**
   - For F, click the check box, and then type **60**.
   - For T_d, click the check box, and then type **0**.
   - For θ , click the check box, and then type **0**.
   - For ϕ , click the check box, and then type **0**.
   - Click Apply.
4. In the Object page, select **I2**.
5. In the Waveform page, make the following modifications:
   - In the V_a text box, type **2892.31**
   - For F, click the check box, and then type **60**.
   - For T_d, click the check box, and then type **0**.
   - For θ , click the check box, and then type **0**.
   - For ϕ , click the check box, and then type **120**.
   - Click Apply.
6. In the Object page, select **I3**.
7. In the Waveform page, make the following modifications:
   - In the V_a text box, type **2892.31**
   - For F, click the check box, and then type **60**.
   - For T_d, click the check box, and then type **0**.
   - For θ , click the check box, and then type **0**.
   - For ϕ , click the check box, and then type **240**.
   - Click Apply.
8. On the File menu, click Save.
9. Close the Circuit window.

Step 5: Generating the Time-Harmonic field solution

Set the solving options

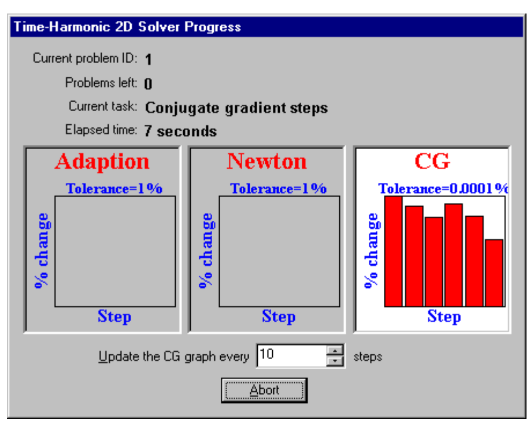


1. On the Solve menu, click Set Solver Options
2. In the Set Solver Options dialog, select **2** as the Polynomial order value, select Non-linear as the Material type, and accept all other default values:
   - Method: **Newton-Raphson**
   - Maximum Newton iterations: **20**
   - Newton tolerance: **1**%
   - CG tolerance: **0.01**%
   - Source frequency: **60 Hertz**
3. Click OK.

Solve the model

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

  The Time-Harmonic 2D Solver Progress dialog appears.



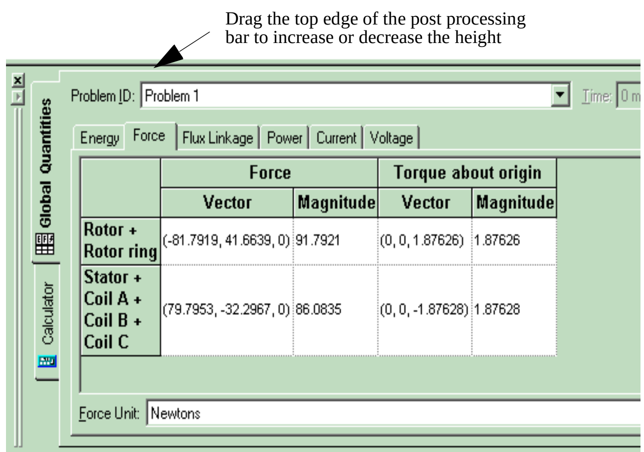
Step 6: Analyzing the results for locked rotor analysis

In this section, we will examine the post-processing toolbar and check the value of the torque for the body that is made of the rotor and rotor ring - this is the locked rotor torque. The result should be 3.826 N/m, however, since only half of the device is modeled, the torque that the program computes is half the value, i.e., 1.913 N/m.

View the torque

The Post Processing bar is automatically displayed at the bottom of the main window when the Solver Progress dialog closes and the solution is complete. To view the torque, open the Force page.

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.

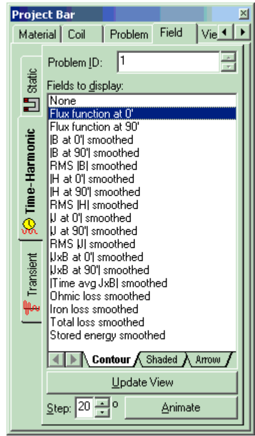


Note The results should be 1.913 N/m, and not 1.87626 N/m, which means that there is an error of about 1.9 %. Further refinement is needed to get a more accurate torque value.

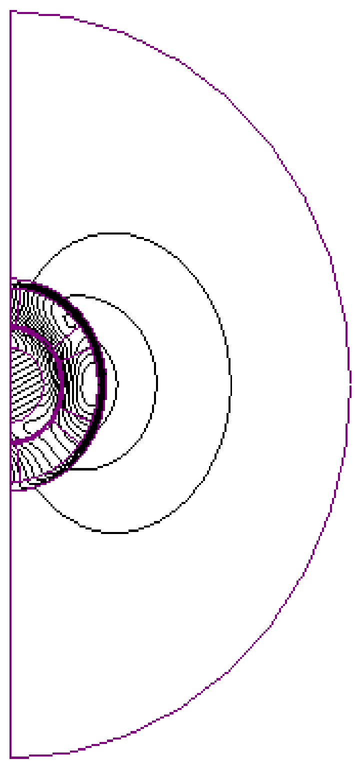
View the contour plot

The contour plot displays contour lines of the magnetic flux function.

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



2. In the Fields to Display list, make sure that **Flux Function at 0^0** is selected.
3. At the bottom of the Field page, click Update View.
   The contour plot should look similar to the illustration shown below.

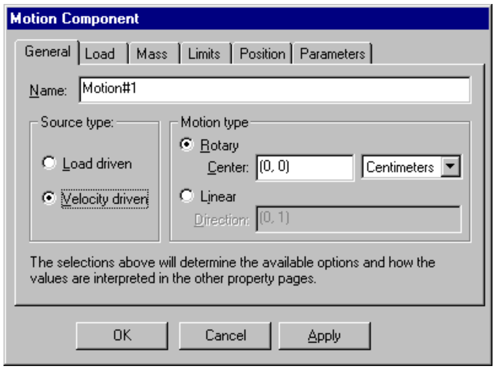


Step 7: Setting up the problem for analysis of torque at a speed of 200 rad/s

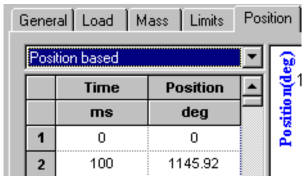
It is possible to use the time harmonic solver, whenever the velocity is constant, by adjusting the frequency of the supply to be the slip frequency. For this part of the tutorial, however, the 2d transient solver is used to solve the problem when the rotor is rotating at a constant velocity of 200 rad/s.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

Making the motion component

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



1. On the View menu, click Solid Model.
2. In the Object page, keeping the CTRL key pressed, click on the Rotor, Rotor ring, and Inner air gap components.
   All three components are selected.
3. On the Model menu, click Make Motion Component.
   The Motion Component dialog appears.
4. In the General page, set the Source Type as **Velocity driven** and the Motion Type as **Rotary**.
5. Open the Position page, select *Position based* from the drop down list, and then enter the following data:
   - Box 1 Time: **0** Position: **0**
   - Box 2 Time: **100** Position: **1145.92**



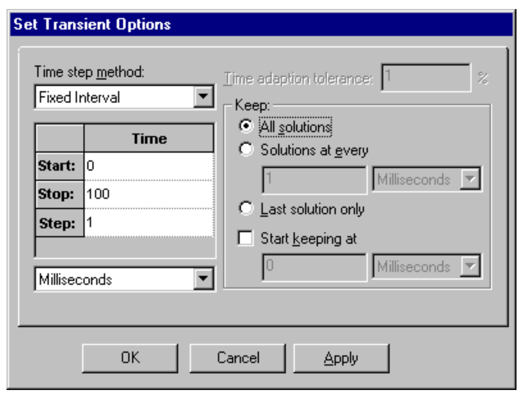
6. Click OK.

Step 8: Generating the Transient 2D with Motion field solution

Set the solving options

1. On the Solve menu, click Set Solver Options, and verify that Polynomial Order value is **2** and that the following default values are set:
   - Material type: **Non-linear**
   - Method: **Newton-Raphson**
   - Maximum Newton iterations: **20**
   - Newton tolerance: **1**%
   - CG tolerance: **0.01**%
   - Source frequency: **60 Hertz**
2. Click OK or Close.

Set the Transient options

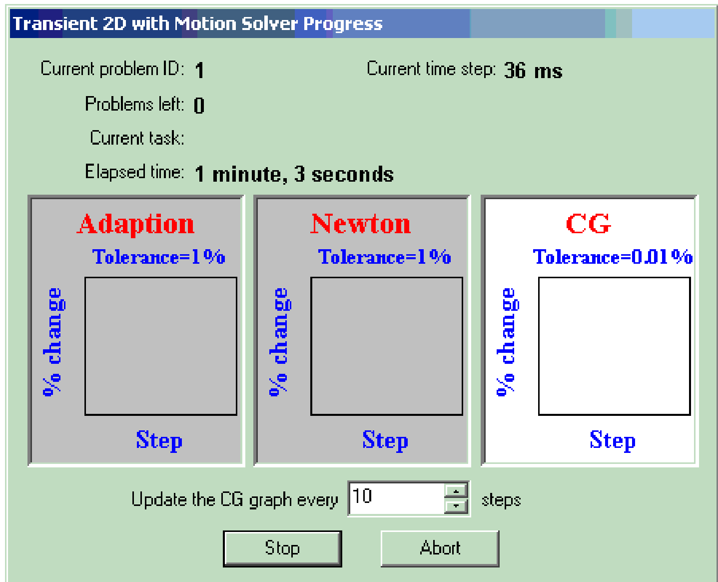


1. On the Solve menu, click Set Transient Options.
   The Set Transient Options dialog appears.
2. Make sure that *Fixed Interval* is selected as the Time Step Method, and then make the following modifications for Time:
   - Start = **0** Milliseconds
   - Stop = **100** Milliseconds
   - Step = **1** Milliseconds
3. Click OK.

Solve the model

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

  The Transient 2D with Motion Solver Progress dialog appears.



Note This tutorial was prepared using a Pentium 266 processor with 128mb of RAM. The 100 steps of this solution took approximately five minutes to complete. Time may vary, depending on the speed and memory of your computer.

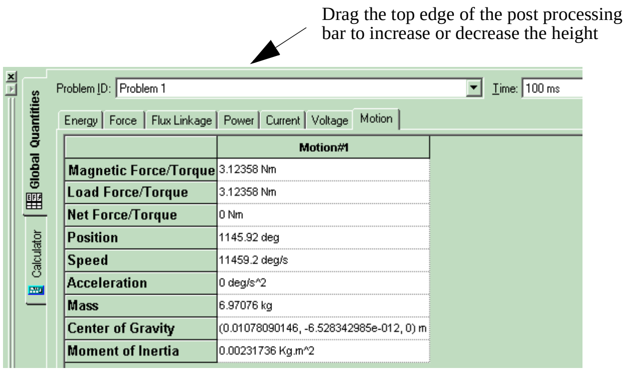
Step 9: Analyzing the results for torque at a speed of 200 rad/s

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

Graph the magnetic torque

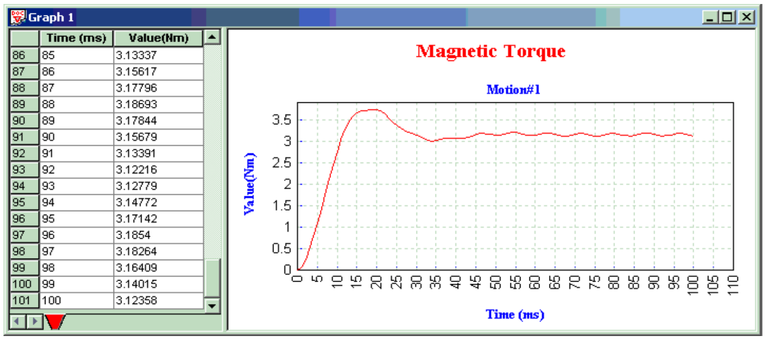
The Post Processing bar is automatically displayed at the bottom of the main window when the Solver Progress dialog closes and the solution is complete. To view the magnetic torque, open the Motion page.

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. In the Post Processing bar, open the Motion page.
2. Select **100 ms** from the Time drop down list.
3. Using the mouse pointer, click anywhere inside the Magnetic Force/Torque text box (3.12358 Nm).
4. Click the Graph Selection button, located at the upper right corner of the Post Processing bar.

   The graph should look like the illustration below.



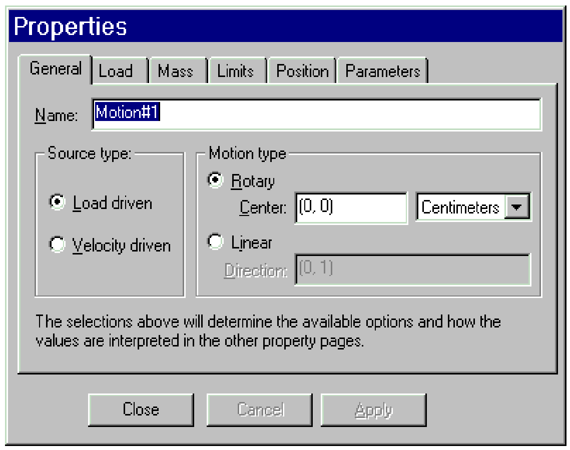
5. On the File menu, click Save.

Step 10: Setting up the problem for analysis of the start-up condition

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

Modifying the motion component

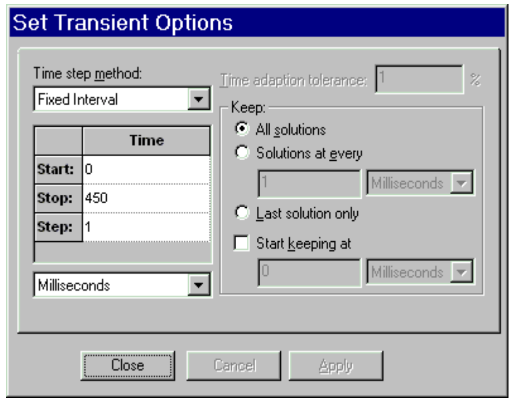
For the purpose of this analysis, we will need to set the motion component to be load driven. This means that the velocity of the motion component is not known and that it varies with time as a result of the generated electromagnetic field.



1. In the Object page, click on the Motion#1 component.
2. On the Edit menu, click Properties.
   The Motion Component dialog appears.
3. In the General page, set the Source Type as **Load driven**.
4. Click OK.

Modifying the Transient options

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

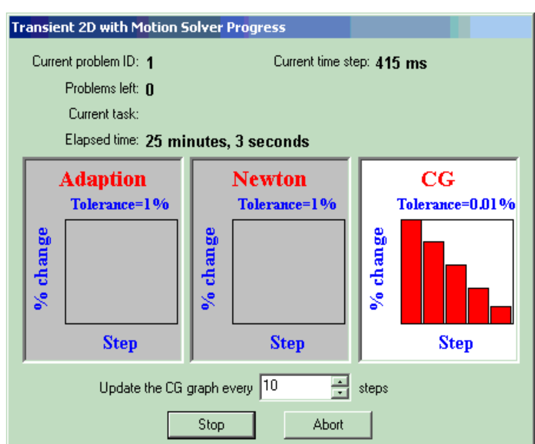


1. On the Solve menu, click Set Transient Options.
   The Set Transient Options dialog appears.
2. Make sure that *Fixed Interval* is selected as the Time Step Method, and then make the following modification for Stop Time:
   - Start = **0** Milliseconds
   - Stop = **450** Milliseconds
   - Step = **1** Milliseconds
3. Click OK.

Solve the model

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

  The Transient 2D with Motion Solver Progress dialog appears.



Note This tutorial was prepared using a Pentium 266 processor with 128mb of RAM. The 450 steps of this solution took approximately 53 minutes to complete. Time may vary, depending on the speed and memory of your computer.

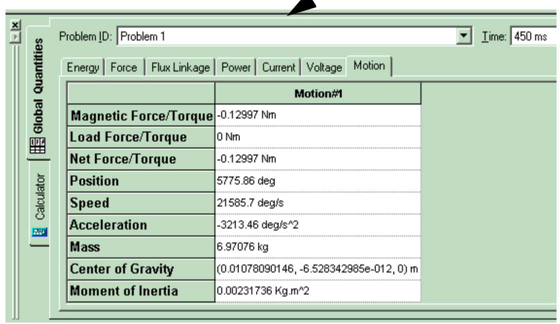
Step 11: Analyzing the results for rotor speed and torque during start-up

Graph the speed and magnetic torque

The Post Processing bar is automatically displayed at the bottom of the main window when the Solver Progress dialog closes and the solution is complete. Once the solution process is complete, it is possible to examine the speed and magnetic torque of the rotor over the entire time, from standstill to synchronous speed.

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.

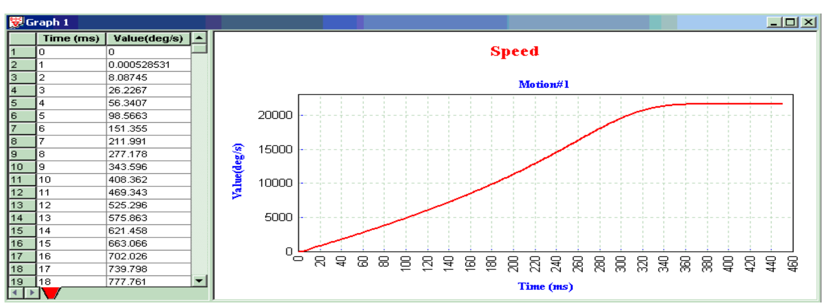
Drag the top edge of the post processing bar to increase or decrease the height



1. In the Post Processing bar, open the Motion page.
2. Select **450 ms** from the Time drop down list.
3. Using the mouse pointer, click anywhere inside the Speed text box (21585.7 deg/s).

   Note This is equivalent to 377 rad/s, which is expected from this device.
4. Click the Graph Selection button, located at the upper right corner of the Post Processing bar.

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

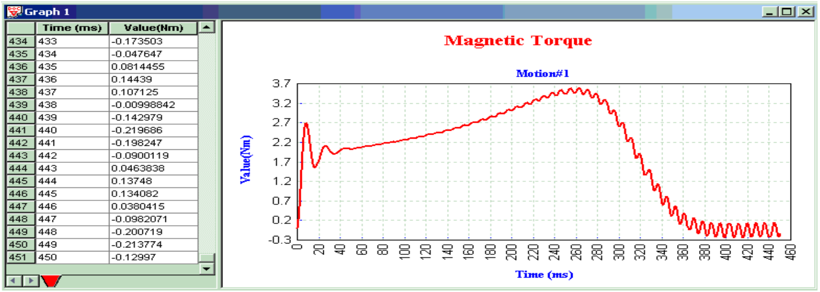


5. Using the mouse pointer, click anywhere inside the Magnetic Force/Torque text box (-0.12997 Nm).

Note At 450 ms, the torque should be zero. The value (-0.12997 Nm) is an indication of numerical and round-off errors.

6. Click the Graph Selection button.

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



7. On the File menu, click Save.

8. On the File menu, click Close.

Summary

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

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

References

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