Ansys 2025 R2

POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT

Getting Started with Maxwell®: Transient Problem

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
https://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2025 R2
July 2025

Conventions Used in this Guide

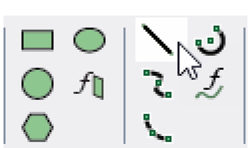
Please take a moment to review how instructions and other useful information are presented in this documentation.

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Bold type is used for the following:
    - Keyboard entries that should be typed in their entirety exactly as shown. For example, “**copy file1**” means you must type the word **copy**, then type a space, and then type **file1**.
    - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by greater than signs (>). For example, “click **Maxwell 3D** > **Excitations** > **Assign** > **Wave Port**.”
    - Labeled keys on the computer keyboard. For example, “Press **Enter**” means to press the key labeled **Enter**.
- Italic type is used for the following:
    - Emphasis
    - Titles of publications
    - Keyboard entries when a name or a variable must be typed in place of the words in italics. For example, “**copy** *filename*” means you must type the word **copy**, then type a space, and then type the name of the file.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, “Press Shift+F1” means to press the **Shift** key and, while holding it down, press the **F1** key also. You should always depress the modifier key or keys first (for example, Shift, Ctrl, Alt, or Ctrl+Shift), continue to hold it/them down, and then press the last key in the instruction.

Accessing Commands: *Ribbons, menu bars,* and *shortcut menus* are three methods that can be used to see what commands are available in the application.

- The *Ribbon* occupies the rectangular area at the top of the application window and contains multiple tabs. Each tab has relevant commands that are organized, grouped, and labeled. An example of a typical user interaction is as follows:

  "Click **Draw** > **Line**"



This instruction means that you should click the **Line** command on the **Draw** ribbon tab. An image of the command icon, or a partial view of the ribbon, is often included with the instruction.

- The *menu bar* (located above the ribbon) is a group of the main commands of an application arranged by category such File, Edit, View, Project, etc. An example of a typical user interaction is as follows:

  "On the **File** menu, click the **Open Examples** command" means you can click the **File** menu and then click **Open Examples** to launch the dialog box.

- Another alternative is to use the *shortcut menu* that appears when you click the right-mouse button. An example of a typical user interaction is as follows:

  “Right-click and select **Assign Excitation** > **Wave Port**” means when you click the right-mouse button with an object face selected, you can execute the excitation commands from the shortcut menu (and the corresponding sub-menus).

Getting Help: Ansys Technical Support

For information about Ansys Technical Support, go to the Ansys corporate Support website, http://www.ansys.com/Support. You can also contact your Ansys account manager in order to obtain this information.

All Ansys software files are ASCII text and can be sent conveniently by e-mail. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached, including software files as applicable. This allows more rapid and effective debugging.

Help Menu

To access help from the Help menu, click **Help** and select from the menu:

- **[product name] Help** - opens the contents of the help. This help includes the help for the product and its *Getting Started Guides*.
- **[product name] Scripting Help** - opens the contents of the *Scripting Guide*.
- **[product name] Getting Started Guides** - opens a topic that contains links to Getting Started Guides in the help system.

Context-Sensitive Help

To access help from the user interface, press **F1**. The help specific to the active product (design type) opens.

You can press **F1** while the cursor is pointing at a menu command or while a particular dialog box or dialog box tab is open. In this case, the help page associated with the command or open dialog box is displayed automatically.

Table of Contents

1 - Introduction

This Getting Started Guide is written for Maxwell beginners and experienced users who would like to quickly refamiliarize themselves with the capabilities of Maxwell. This guide leads you step-by-step through solving and analyzing the results of a rotational actuator magnetostatic problem with motion.

By following the steps in this guide, you will learn how to perform the following tasks:

- Modify a model's design parameters
- Assign variables to a model's design parameters
- Specify solution settings for a design
- Validate a design's setup
- Run a Maxwell simulation
- Plot the magnetic flux density vector
- Include motion in the simulation

> **Note:** This guide assumes that you have already completed the magnetostatic example in *Getting Started with Maxwell: Designing a Rotational Actuator*. If you have not, you may use the project archive in the example directory; however, it is strongly recommended that you complete the magnetostatic example.

Maxwell Solution Types

Ansys Maxwell is an interactive software package that uses finite element analysis (FEA) to simulate (solve) electromagnetic field problems. Maxwell integrates with other Ansys Electromagnetic software to perform complex tasks while remaining simple to use. Maxwell incorporates a set of 2D and 3D solvers in the Ansys Electronics Desktop integrated user interface. This guide will focus on 3D capabilities. A 2D problem example is covered in a separate Getting Started Guide.

The following six types of stand-alone solutions are supported by Maxwell 3D:

- **Magnetostatic linear and nonlinear 3D fields** caused by a user-specified distribution of DC current density and permanent or externally applied magnetic fields. Materials can be nonlinear and anisotropic. Additional quantities that can be computed include torque, force, and self and mutual inductances.
- **Harmonic (sinusoidal variation in time) steady-state magnetic fields** with pulse-induced eddy currents in massive solid conductors caused by one of the following:
  - A user-specified distribution of AC currents (all with the same frequency but with possibly different initial phase angles).

- Externally applied magnetic fields.

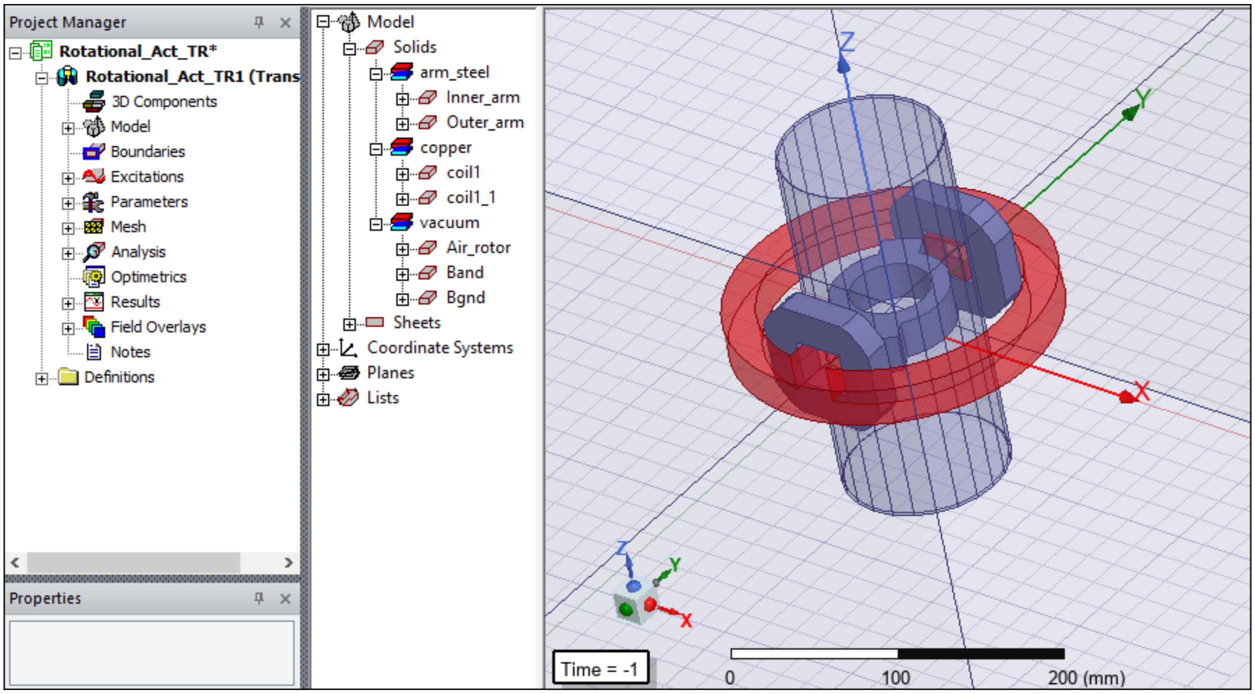
  This solution includes displacement currents for calculating near field electromagnetic wave radiation.

- **Transient (time domain) magnetic fields** caused by permanent magnets, conductors, and windings supplied by voltage and/or current sources with arbitrary variation as functions of time. Rotational or translational motion effects can be included in the simulation. This option uses the TAU solver.
  - **A-Phi Formulation**: This solver is an alternative to the transient TAU solver. *A* is the Magnetic Vector Potential and ϕ(Phi) is the Electric Scalar Potential. The A-Phi solver allows multi-terminal conductors, and it supports sources of various types on a single conduction path; the J and E fields are calculated directly because it is a first-order approximation. This option is useful for application fields in Power Electronics, Bus Bars, PCBA, *etc.*, where the voltage distribution in multiple conductors is essential.
- **Electrostatic 3D fields** caused by a user-specified distribution of voltages and charges in nonconducting regions. Additional quantities that can be computed include torque, force, and capacitances.
- **Electric DC conduction 3D fields** in conductors characterized by a spatial distribution of voltage, electric field, and current density. Power loss can also be computed. In addition, optional simulation of fields in insulating materials is supported.
- **Transient (time domain) 3D electric fields** caused by time dependent voltage, current and charge distributions. All sources are arbitrary functions of time.
- **AC Conduction** 3D electric fields and losses arising in conductors and imperfect (lossy) dielectrics from the application of an alternating (AC) voltage or external current to the electrodes. The AC conduction field solver assumes that all sources are sinusoids oscillating at the same frequency.

In addition, Maxwell may be coupled with other simulators to provide a greater range of solution capability. Couplings to Workbench for thermal and stress analysis, HFSS for ferrite analysis, and Twin Builder for Finite Element/Circuit cosimulation are all supported.

The Maxwell Desktop

The following graphic shows the different sections of the Maxwell desktop:



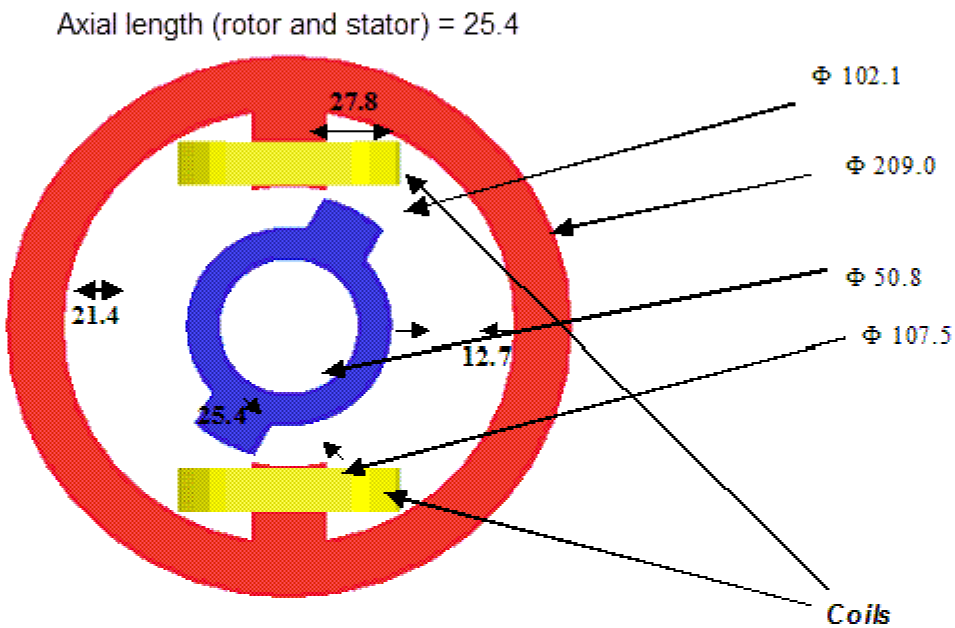
General Procedure for Setting Up Maxwell Designs

You are not required to follow a specific order when setting up your Maxwell design. However, the following order is recommended, particularly for new users:

1. Open Ansys Electronics Desktop by double-clicking the desktop icon or by clicking **Start** > **Programs** > **Ansys EM Suite** [version] > **Ansys Electronics Desktop** [version] from the Windows taskbar.
2. Add a Maxwell 3D design and save the new project.
3. Draw the geometry of the model.
4. Optionally, modify the model's design parameters.
5. Assign variables to design parameters.
6. Assign excitations and boundary conditions.
7. Specify solution settings.
8. Run a Maxwell simulation.
9. Create post-processing plots.
10. Create a parametric analysis.
11. Create a field animation of the parametric analysis results.
12. Include motion in the transient design.

About the Example Design

The application described in this Getting Started guide is an extension of the design created in *Getting Started with Maxwell: Designing a Rotational Actuator.* The geometry is shown below:



The outer part is a nonlinear, ferromagnetic armature carrying two coils. The inner part is made of the same nonlinear material and can rotate around an axis. The inner and outer parts of the device are coaxial.

The field distribution will likely cause the flux density to concentrate in the two steel armatures in the regions where the distance between them is minimal. The expected edge effect will then further increase the field concentration.

In this example, we will compute the torque acting on the inner armature and the flux linkage of the two coils. Simulation results show a 3D electromagnetic time-transient problem with the effects of large motion included. Both the rotor and stator are made of solid ferromagnetic steel, creating significant eddy current effects. A nonlinear B-H curve is considered for the stator and rotor steel. The solution includes the estimated mechanical rotor inertia. For a presentation of the results and the corresponding FEM code, see the *IEEE Transactions on Magnetics, Vol 38, No. 2, March 2002, pp 609-612*.

2 - Setting Up the Design

In this chapter, you will complete the following tasks:

- Open and save a new project
- Insert a new Maxwell design into the project
- Select a solution type for the project
- Set the drawing units for the design

Open and Save a New Project

A project is a collection of one or more designs that is saved in a single *.aedt file. A new project is automatically created when Ansys Electronics Desktop is launched.

To create a new Maxwell 3D design project:

1. On Windows, click **Start** > **Ansys EM Suite** 2025 R2 **Ansys Electronics Desktop** 2025 R2.

   On Linux, from the command line, `cd` to your /v252/AnsysEM/ directory, and enter `./ansysedt`

2. Click **Project** > **Insert Maxwell 3D Design**.

   The new design is listed in the Project Manager tree. By default, it is named **Maxwell3DDesign1**. The **Modeler** window appears to the right of the Project Manager window.

3. Click **File** > **Save As**.

   The **Save As** dialog box appears.

4. Locate and select the folder in which you want to save the project.
5. Type **Rotational_Act_TR** in the **File name** box, and click **Save**.

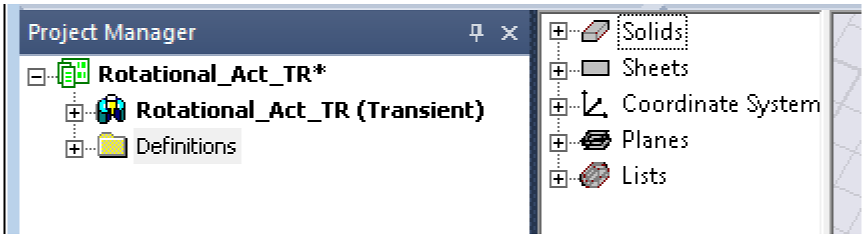
   The project is saved in the specified folder under the name **Rotational_Act_TR.aedt**.

6. Rename the design:
   a. Right-click **Maxwell3DDesign1**.

      A shortcut menu appears.

   b. Select **Rename**.
      The design name becomes highlighted and editable.
   c. Type **Rotational_Act_TR** as the name for the design, and press **Enter**. The project

and design are now both named **Rotational_Act_TR**.



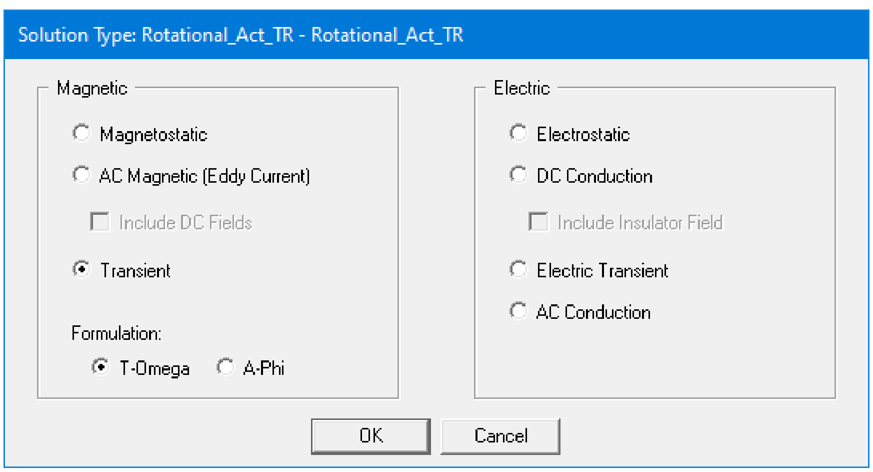
Specify a Solution Type

As mentioned in the introduction, multiple solution types are available, depending on the specific application. For this design, choose a **Transient** solution.

To specify the solution type:

1. Click **Maxwell 3D** > **Solution Type**.

   The **Solution Type** dialog box appears.



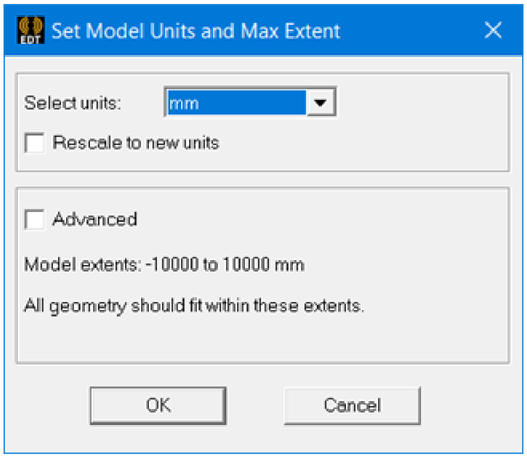
2. Select the **Transient** radio button. Leave the formulation to the default **T-Omega**.
3. Click **OK**.

Set the Drawing Units

To set the drawing units:

1. Click **Modeler** > **Units**.

   The **Set Model Units and Max Extent** dialog box appears.



2. Select **mm** from the **Select units** drop-down menu.
3. Click **OK**.

3 - Importing the Geometric Model

In this chapter you will open the Magnetostatic Getting Started project, copy the objects definitions and material properties, and paste the objects and materials into the **Rotational_Act_TR** transient project.

If you have not completed the magnetostatic guide *Getting Started with Maxwell: Designing a Rotational Actuator*, it is strongly recommended that you do so to gain necessary knowledge of modeling and material assignment principles. You may, however, use the project in the example directory.

In this chapter you will complete the following tasks:

- Open the Magnetostatic Getting Started example.
- Copy and paste geometry and materials to the current project.

Open the Magnetostatic Project

The geometry and materials used in this project are identical to the magnetostatic guide **Rotational_Actuator** project.

To open the magnetostatic project:

1. Click **File** > **Open Examples**.

   The Windows file browser opens to the Examples folder installed with your application.

2. Locate the **Maxwell\Actuators** folder containing the **Rotational_actuator.aedt** project from the magnetostatic guide, *Getting Started with Maxwell: Designing a Rotational Actuator*.
3. Select the file **Rotational_actuator.aedt** and click **Open**.

   The project is opened and is now listed in the Project Manager Window as shown.

Copy and Paste Objects between Projects

The ability to copy and paste objects and their associated material assignments is a useful and time-saving function of the Ansys Electronics Desktop software.

In order to copy objects, all objects must be selected.

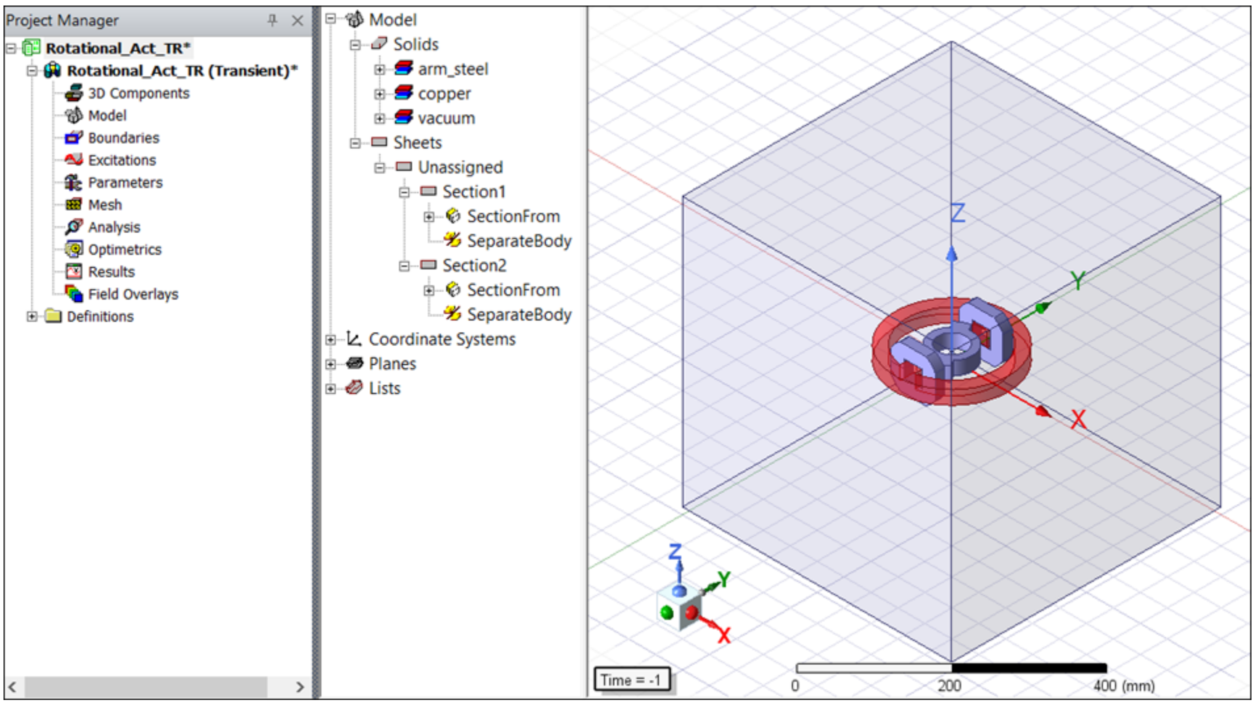
1. With the **Rotational_actuator** project in the modeler window, click in the modeler window and then click **Edit** > **Select All** to select all object regardless of their visibility status.

   The history tree**Model** entry will expand and highlight all objects in the design.

2. Click **Edit** > **Copy** to copy the object and material definitions to the clipboard.

3. Click on the **Rotational_Act_TR(Transient)** design in the Project Manager window to switch the **Modeler** window to the transient project.
4. Click **Edit** > **Paste** to paste all objects and material definitions into the transient project.
5. Click **View** > **Fit All** > **All Views** to fit the objects to the window. You may also use the keyboard shortcut **Ctrl-D**.
6. In the Project Manager window, select the magnetostatic project **Rotational_Actuator** and click **File** > **Close**.

   Your screen should look approximately like the one below.



7. Click **File** > **Save** to save the model before moving on to the next chapter.

4 - Defining the Design Properties

For the transient problem, you want to use a pulse excitation to drive the coils. In order to accomplish this, you will assign a external current winding excitation to the coils and use the Maxwell Circuit Editor to create the external driving circuit including a pulse source. You will also allow the software to calculate the eddy currents in the solid metal objects in the model.

In this chapter you will complete the following tasks:

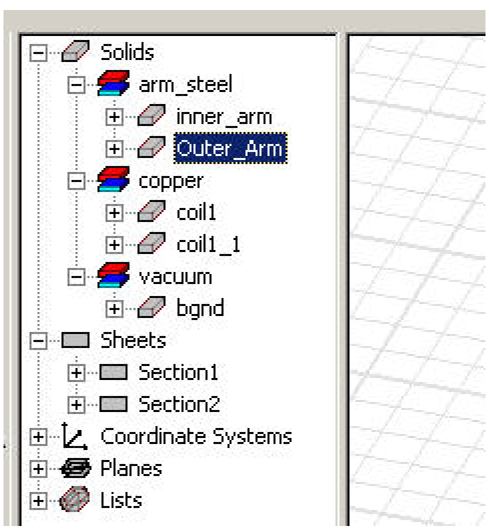
- Verify material properties
- Assign excitations
- Set up an external circuit for the current winding
- Set up the mesh operations
- Specify the eddy effect

Verify Material Properties

Material properties are automatically transferred when you copied the geometry objects. You can view these properties by viewing the **Attribute** tab of the **Properties** window.

To verify the nonlinear material for the armatures:

1. Expand the **Model\Solids** entry in the history as shown.



2. Double-click the **Outer_arm** object.

   The **Properties** window appears.

3. In the **Material** row, click the button in the **Value** column labeled **arm_steel**, then click **Edit**.

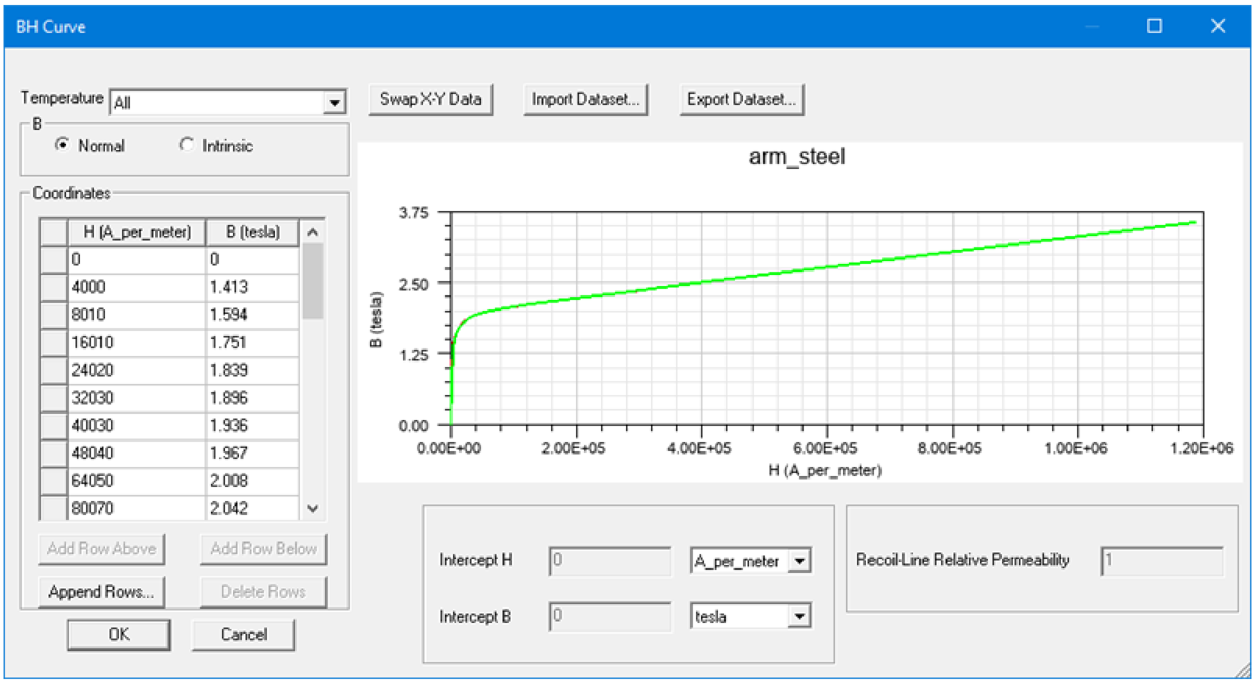
The **Select Definition** dialog box appears.

4. Click the **View/Edit Material** button.

   The **View/Edit Material** dialog box appears.

5. In the **Relative Permeability** row, click the **B-H Curve** button.

   The **BH Curve** dialog box appears as shown.



6. Click **Cancel** in the **BH Curve** dialog box to close.
7. Continue dismissing dialog boxes until you have returned to the **Modeler** window.

Define the Currents

To define the currents:

1. Select **Section1** and **Section 2** under the **Sheets** entry in the history tree.
2. Click **Maxwell 3D** > **Excitations** > **Assign** > **Coil Terminal**.

   The **Coil Terminal Excitation** dialog box appears.

3. Type **350** in the **Number of Conductors** box.
4. Click **OK**.

Assign Excitations

Currents need to be defined and assigned as excitations for the two coil terminals.

Add a Winding

To add a winding for the excitation:

1. Click **Maxwell 3D** > **Excitations** > **Add Winding**.

   The **Winding** dialog box appears.

2. Type **currentwinding** in the **Name** box (the default is **Winding1**).
3. Set the **Type** to **External**.
4. Select the **Stranded** radio button.
5. Leave the **Initial Current** set to the default value of **0** (zero).

   > **Note:** We are using an external circuit to supply the excitation to the coil. For this example, we also could have used a voltage type of excitation.

6. Click **OK**.

Add a Winding Terminal

To add a terminal for the winding:

1. In the Project Manager tree, under **Excitations**, right-click **currentwinding**.

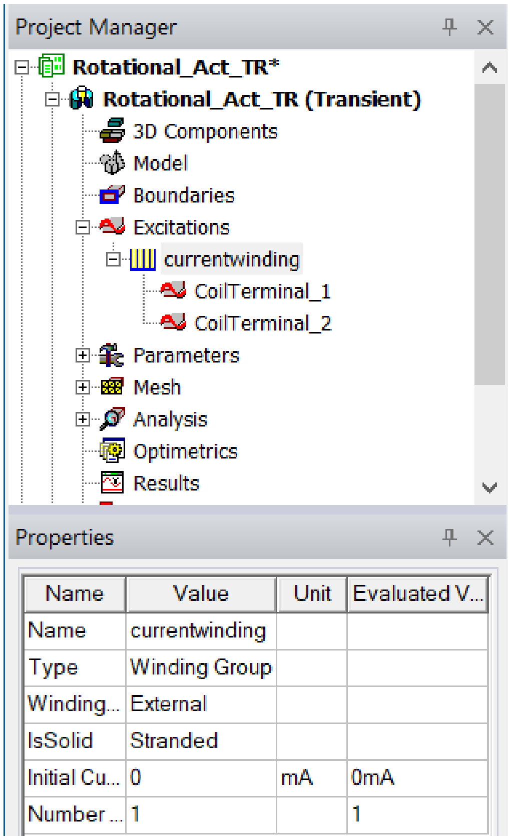
   A shortcut menu appears.

2. Select **Add Terminals** from the shortcut menu.

   The **Add Terminals** dialog box appears.

3. In the list, select **CoilTerminal_1**, press and hold down the **Shift** key, and select **CoilTerminal_2**.
4. Click **OK**.

   In the Project Manager tree, the two terminals are moved beneath the winding as shown below.

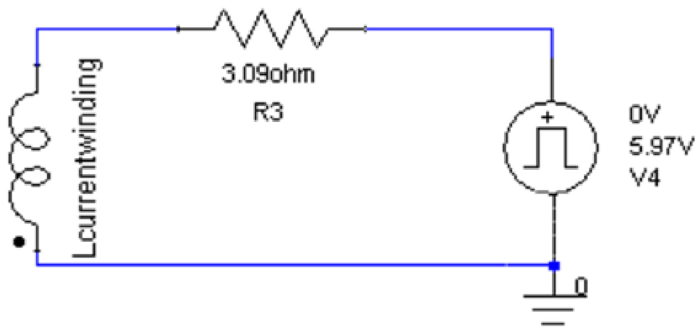


Boundary Conditions

The region box (**Bgnd**) by default has all faces assigned with magnetic flux tangent boundary conditions. Thus, for this problem no additional boundary conditions are needed.

Set Up the External Circuit

The driving circuit for the winding in this design consists of a voltage source in series with a resistor and with the winding. When complete, the circuit should look similar to the following figure:

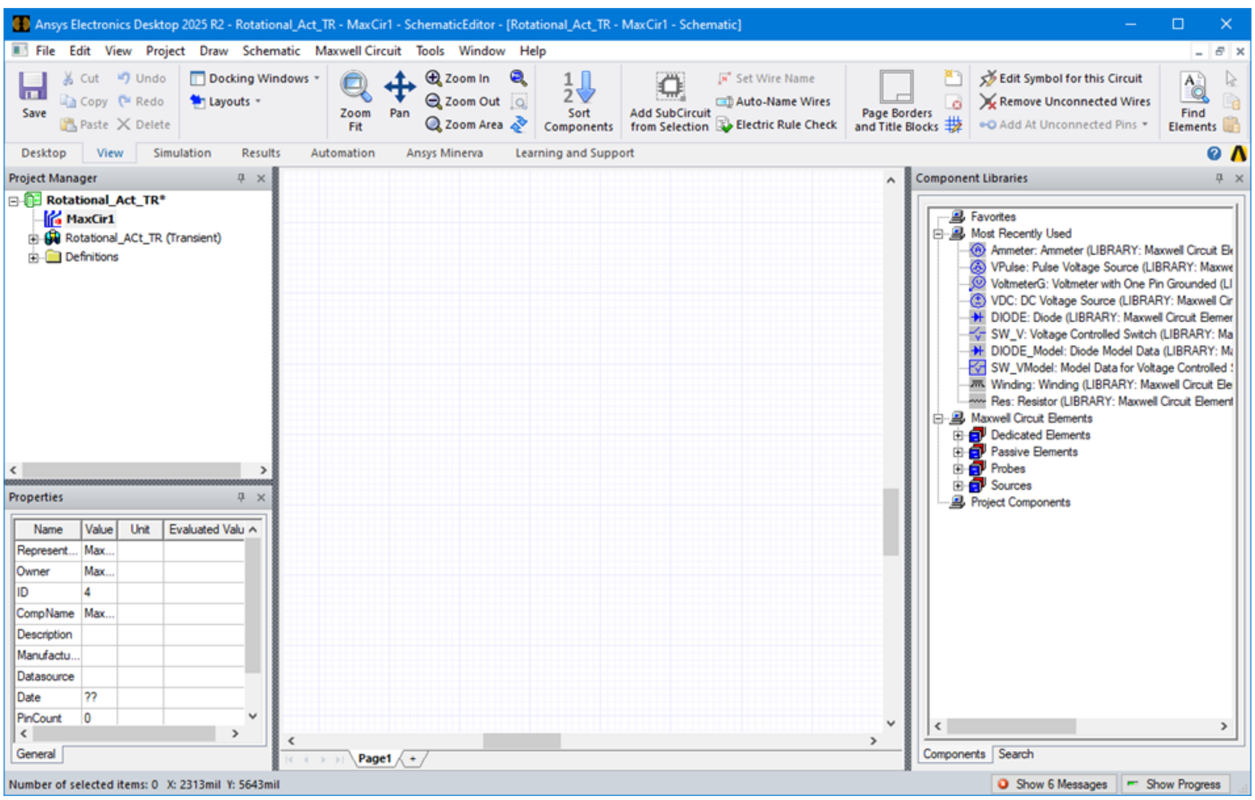


Add the Circuit Elements

To add the circuit elements in Maxwell Circuit Editor:

1. Click **Project** > **Insert Maxwell Circuit Design**.

   The Maxwell Circuit Editor opens with a default circuit sheet as shown below.



2. Click the **Components** tab in the Component libraries window.
3. Place the winding circuit element on the sheet:

   a. In the Component Libraries tree, under **Maxwell Circuit Elements/Dedicated Elements**, select the **Winding** element.
   b. Click on, and drag the **Winding** element onto the sheet.
   c. Right-click in the Schematic window, and select **Finish** to exit component placement mode.
   d. To view the properties, double-click the component in the Schematic window.

      The **Properties** window appears.

   e. Change the **Name** to **currentwinding**, the same name you used when defining the winding in the Maxwell design.
   f. Click **OK**.
   g. Click **Draw** > **Rotate**, and position the winding vertically.
4. Place a resistor on the sheet:
   a. In the Component Libraries tree, under **Passive Elements**, select **Res:Resistor**.
   b. Drag the resistor onto the sheet.
   c. Right-click, and select **Finish** to exit placement mode.
   d. Double-click the symbol of the resistor, change the value of the resistor, **R**, to **3.09**, keep the **Unit** value set to **ohm**, and click **OK**.
5. Place a voltage pulse on the sheet:
   a. In the Component Libraries tree, under **Sources** select a **VPulse** element (Pulse Voltage Source).
   b. Drag it to the sheet, and then right-click and select **Finish**.
   c. Double-click the source element symbol on the sheet, and then specify the following source characteristics:

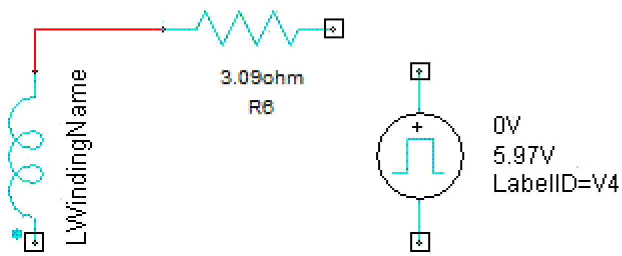
Parameter	**Value**	**Description**
V1	0	Initial voltage
V2	5.97	Peak voltage
Td	0	Initial delay time
Tr	0.001	Rise time
Tf	0.001	Fall time
Pw	1	Pulse width
Period	2	

   d. Leave the other fields set to the default values, and click **OK**.

Connect the Circuit Elements in Series

To connect the circuit elements in series:

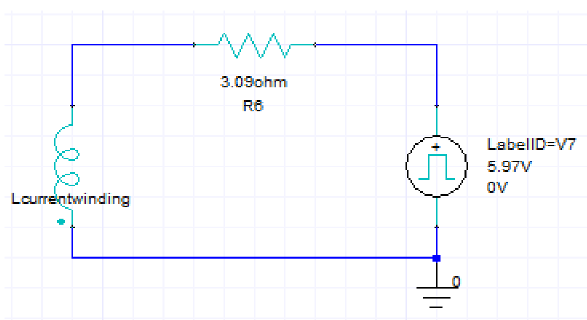
1. From within the Maxwell Circuit Editor, click **Draw** > **Wire**.
2. Click one terminal of the inductor and draw the wire to one resistor terminal as shown.



3. Repeat until a closed circuit is formed.

> **Note:** For the current winding, the "dotted" terminal is positive and current is positive when it flows from the "dotted" terminal to the "undotted" terminal. Connecting the winding as shown results in negative current in the plot in the *Create a Current vs Time Plot* chapter. You may reverse the winding orientation with no change to the problem except for the plot, which would show positive current in that case.

4. When done, place the **Ground** symbol:
   a. Click **Draw** > **Ground**.
   b. Place the **Ground** symbol on the sheet as shown below, right-click, and select **Finish**.



Export the Netlist

To export the netlist:

1. From within the Maxwell Circuit Editor, click **Maxwell Circuit** > **Export Netlist**.

   The **Netlist Export** dialog box appears.

2. Select the folder where you want to save the external circuit file.
3. Type **trans_circ** in the **File name** box.
4. Click **Save** to save the **trans_circ.sph** file.

   The **Netlist Export** dialog box closes and the Maxwell Circuit Editor reappears.

Save the Maxwell Circuit Design

To save the Maxwell Circuit design:

1. Right-click the circuit design in the Project Manager tree, select **Rename** on the context menu, and enter **ExternalCircuit** as the name.
2. Click **File** > **Save** to save the project with the new circuit design.

Assign the External Circuit

To assign the circuit in Maxwell:

1. Click **Maxwell 3D** > **Excitations** > **External Circuit** > **Edit External Circuit**.

   The **Edit External Circuit** dialog box appears.

2. Click **Import Circuit Netlist**.

   The **Select File** dialog box appears.

3. Select **Maxwell Circuit Netlist Files (*.sph)** from the **Files of type** drop-down menu.
4. Browse to the location where you saved the circuit, select **trans_circ.sph**, and click **Open** to import it.
5. Click **OK** to close the **Edit External Circuit** dialog box.

Set Up Mesh Operations

This example involves transient magnetic fields in the presence of massive (solid) conductors, resulting in eddy currents. To catch the effects with reasonable accuracy, a finer mesh is required in those objects because skin effects are part of an accurate transient solution.

To seed the mesh to the desired density in the **Outer_arm** and **Inner_arm** objects:

1. Select **Outer_arm** from the history tree, press and hold down **Ctrl**, and then select **Inner_arm**.
2. Click **Maxwell 3D** > **Mesh** > **Assign Mesh Operation** > **Inside Selection** > **Length Based**.

   The **Element Length Based Refinement** dialog box appears.

3. Type **10** in the **Maximum Length of Elements** box, and select **mm** as the units.

4. Leave the **Maximum number of additional elements** check box unchecked.
5. Click **OK**.

This operation refines the mesh at run-time before the transient problem solution begins. This mesh will be used for all time steps; therefore, the mesh density should be appropriate for the anticipated field behavior for the entire transient analysis.

Specify the Eddy Effect Calculation

Eddy effects can be calculated in objects with non-zero electric conductivity.

To calculate eddy effects:

1. Click **Maxwell 3D** > **Excitations** > **Set Eddy Effects**.

   The **Set Eddy Effect** dialog box appears.

2. Select the check boxes for the **Inner_arm** and **Outer_arm** objects.
3. Click **OK**.

Specify Torque Calculations

To set up the torque calculation:

1. Select the **Inner_arm** object in the history tree.
2. In the Project Manager tree, right-click **Parameters** row.

   A shortcut menu appears.

3. Select **Assign** > **Torque** from the shortcut menu.

   The **Torque** dialog box appears.

4. Select **Global:Z** from the **Axis** drop-down list.
5. Select the **Positive** radio button for the axis orientation.
6. Click **OK**.

5 - Setting Up and Running the Analysis

In this chapter you will complete the following tasks:

- Set up the analysis
- Run and solve the analysis

Set Up the Analysis

To set up the analysis:

1. Right-click **Analysis** in the Project Manager tree, and select **Add Solution Setup**.

   The **Solve Setup** dialog box appears.

2. Click the **General** tab.
3. Type **0.04** in the **Stop time** box, and select **s** as the unit.
4. Type **0.005** in the **Time step** box, and select **s** as the unit.
5. To save fields during the solution:
   a. Click the **Save Fields** tab.
   b. Select **Custom**. A row is added to the table that should have the following values:
      - **Linear Step**. This type-of-sweep setting can also be selected from the Distribution drop-down list.
      - **0.0** in the **Start** box.
      - **0.04** in the **End** box.
      - **0.005s** in the **Step size** box.

6. Click **OK**.
7. Click **File** > **Save** to save the model before running the analysis.

Validate Design

Before running a simulation, it is best to validate its setup.

1. From the **Maxwell 3D** menu, select **Validate Check**. Alternatively, from the **Simulation** tab, click the **Validate** icon. 

2. Address any issues reported in the Validate Check window.

Run the Analysis

To run the analysis:

- Right-click **Analysis** in the Project Manager tree, and select **Analyze** in the shortcut menu. The time required to complete the analysis depends upon the speed and memory capability of your machine and other applications that may be using machine resources. The status of the simulation is reported in the **Progress** bar.

6 - Postprocessing the Results

In this chapter you will complete the following tasks:

- Plot the magnetic flux density vector
- Plot the current density distribution
- Plot the torque versus time
- Plot the current versus time

Plot the Magnetic Flux Density Vector

This section describes how to plot the flux density vector on the mid-vertical symmetry plane of the device. You previously set up a relative coordinate system (CS1) containing the desired plot plane, so now you will do the following:

- Create an Object Selection
- Plot the Quantity

Create an Object Selection

Of particular interest are the results in the two armatures, so to plot only those results, create a list of these two objects to prepare for the plot:

1. Select the **Outer_arm** and **Inner_arm** objects in the **Modeler** tree.
2. Click **Modeler** > **Named Selection** > **Create** > **Object Selection**.

   The list of selected objects (**ObjectSelection1**) is added under **Named Selections** in the **Modeler** tree.

Plot the Quantity

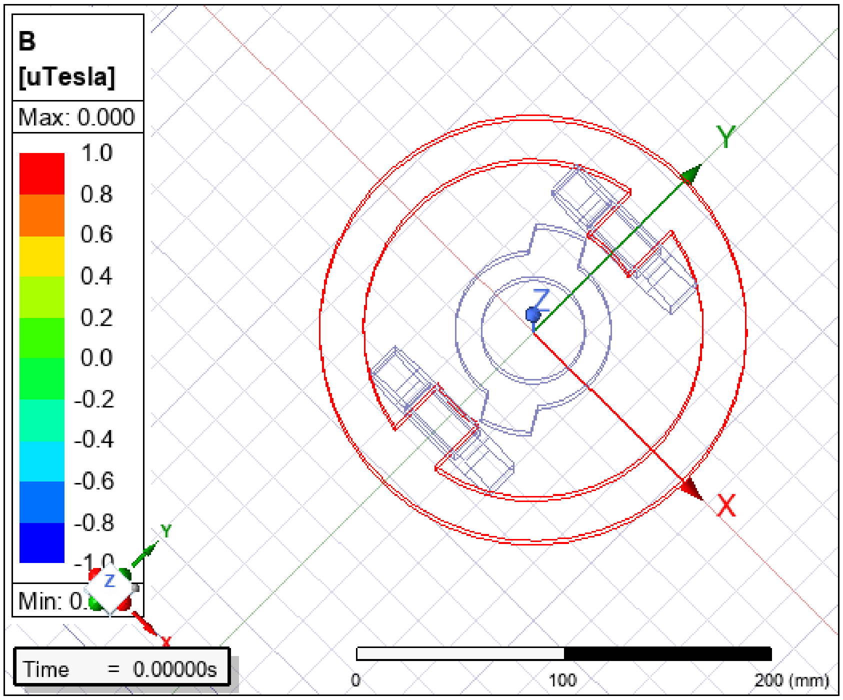
To create the plot:

1. Change the rendering of both **Outer_arm** and **Inner_arm** to wireframe by clicking **View** > **Render** > **Wire Frame**.
2. In the Modeler tree, select the **RelativeCS1:XY** plane under **Planes**.
3. In the Project Manager tree, right-click **Field Overlays**, and select **Fields** > **B** > **B_ Vector**.

   The **Create Field Plot** dialog box appears.

4. Make sure **B_Vector** is selected in the **Quantity** list.
5. Select **ObjectSelection1** in the **In Volume** list.
6. Click **Done**.

7. The **B_Vector** plot is displayed as shown below. The vector values are zero at **0 sec** as a result of the pulse source used in the winding setup having zero initial voltage.



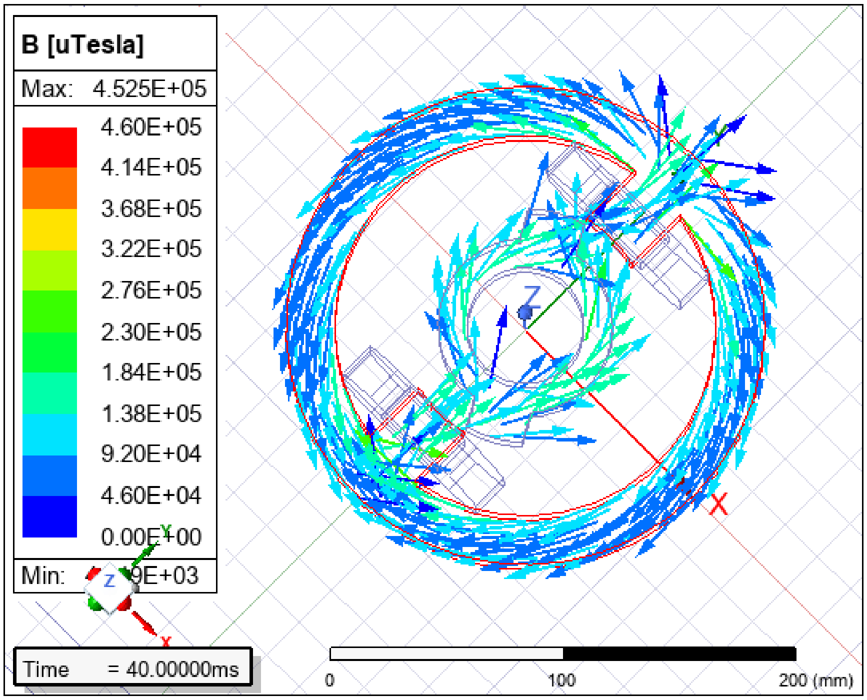
Set the Solution Context

To change the time step for post processing:

1. Click **View** > **Set Solution Context** or double click on the time context display in the modeler window.

   The **Set View Context** dialog box appears.

2. Select the **Setup1:Transient** from the **Solution Name** drop-down list.
3. Set the time step from the **Time** drop-down menu to **40 ms**.
4. Click **OK**. The plot automatically updates to the new time as shown.



5. Right-click on **B_Vector1** in the Project Manager tree, and select **Plot Visibility** to turn off the plot.
6. Set the **Solution Context** back to **0s**.

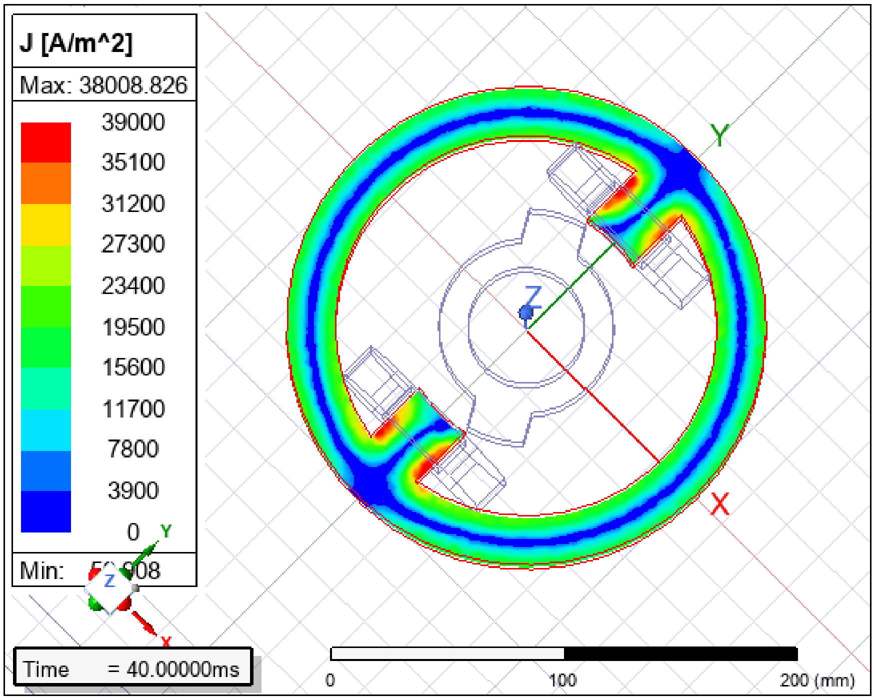
Plot the Current Density Distribution

To plot the current density distribution on the same XY plane of CS1 (**RelativeCS1:XY**):

1. Select the **RelativeCS1:XY** plane.
2. Right-click in the Modeler window, and select **Fields** > **J** > **Mag_J** from the shortcut menu.

   The **Create Field Plot** dialog box appears.
3. Select **Outer_arm** from the **In Volume** List.
4. Click **Done** to plot.
5. Set the **Solution Context** to **40 ms**.

   The field partially penetrates the stator, and the transient distribution of the current density shows significant skin effects.



Plot Torque and Current

An important transient analysis feature is the ability to vary global quantities as a function of time. Examples of such quantities include currents and voltages, power loss, torque/force, flux linkage of windings, and induced voltages.

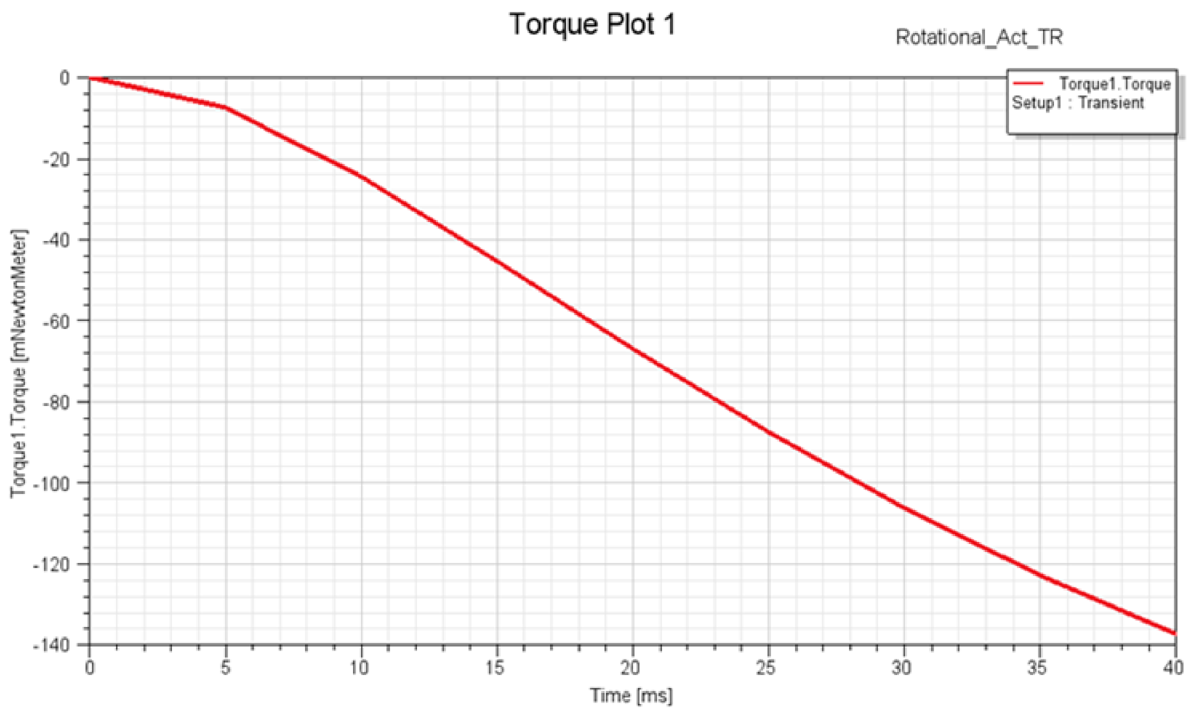
Create a Torque vs. Time Plot

To create a plot of the torque as a function of time:

1. Right-click **Results** in the Project Manager tree, and select **Create Transient Report** > **Rectangular Plot**.

   The **Traces** dialog box appears.

2. From the **Solution** drop-down menu, select the solution setup (**Setup1**).
3. From the **Category** list, select **Torque**.
4. Click the **New Report** button.
5. Click **Close** to dismiss the dialog box.



Create a Current vs. Time Plot

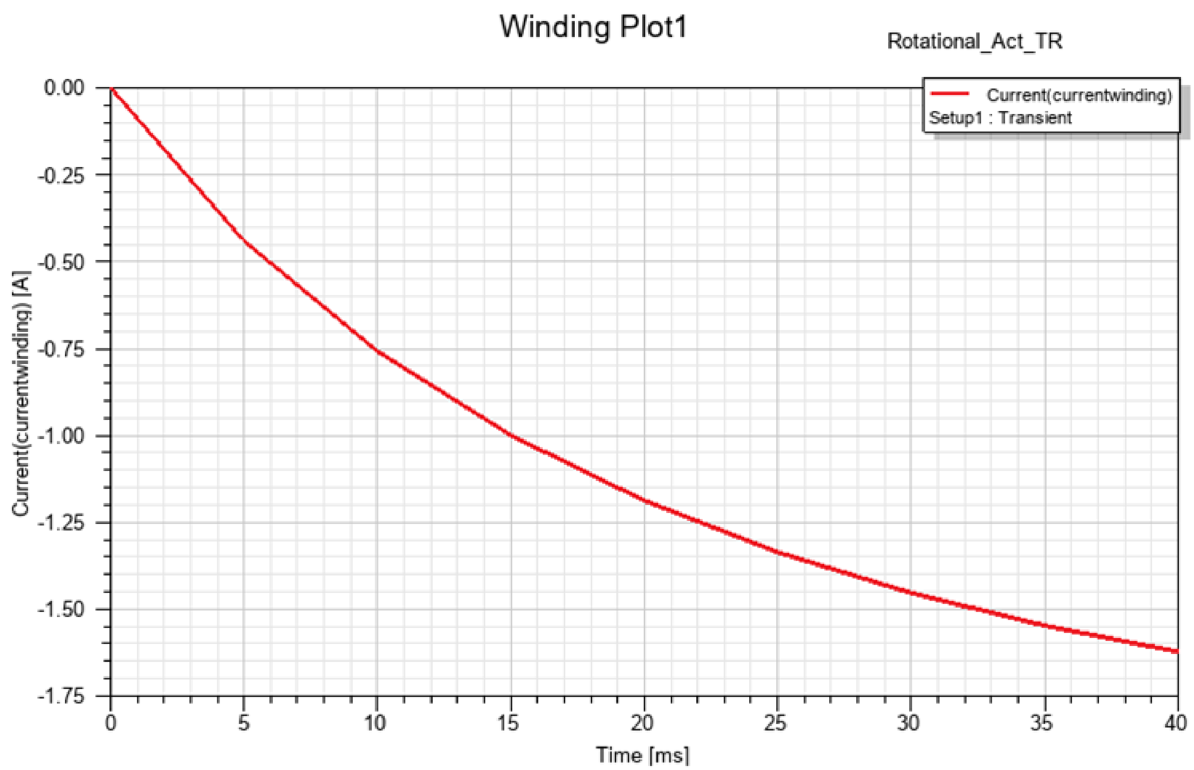
To create a plot of the current as a function of time:

1. Right-click **Results** in the Project Manager tree, and select **Create Transient Report** > **Rectangular Plot**.

   The **Traces** dialog box appears.

2. From the **Solution** drop-down menu, select the solution setup (**Setup1**).
3. From the **Category** list, select **Winding**.
4. From the **Quantity** list, select **Current**.
5. Click the **New Report** button.

6. Click **Close** to display the dismiss the dialog box.



Close the Plot

To close the open plot:

- Click the **X** in the upper right corner of the plot window.

> **Note:** After you close a plot, it is still available to view later, listed under **Results** in the Project Manager tree.

7 - Including Motion in the Simulation

In order to include the effects of motion of the **Inner_arm**, the object must be isolated from the rest of the model using a mesh band. In order to create this mesh band, you will add two objects, between the **Inner_arm** and **Outer_arm** objects.

In general, any moving object must be isolated from the stationary model using a mesh band. For more information, see Meshing and Band Setting Recommendations for 3D Transient Applications with Motion.

In this chapter, you will complete the following tasks:

- Add large motion to the simulation
- Analyze the transient solution with motion
- Post process the transient results
- Close the project and exit Maxwell

Create a New Model

Before adding motion to the design, save the "without motion" (or non-transient) design and create a copy.

To save and copy the design:

1. Click **File** > **Save** to save the design.
2. In the Project Manager tree, right-click the **Rotational_Act_TR** design listed under the project, and select **Copy**.
3. In the Project Manager tree, right-click the name of the project (also **Rotational_Act_TR**), and select **Paste**.

   A second copy of the same design appears under the single project; by default, the name of the new design is **Rotational_Act_TR1**.

4. Double-click **Rotational_Act_TR1** to make it active.

Add a Band Object to the Design

The band object is a regular polyhedron positioned so that it contains all rotating objects inside it.

To add the band object:

1. Set the working coordinate system to CS1:

   Click **Modeler** > **Coordinate System** > **Set Working CS**, select **RelativeCS1**, and click **Select**.

2. Create a regular polyhedron around the Z axis named **band**:
   a. Click **Draw** > **Regular Polyhedron**.
   b. Type **(0, 0, -121)** in the **(X, Y, Z)** boxes, for the origin, and then press **Enter**.
   c. Type **(52.5, 0, 0)** in the **(dX, dY, dZ)** boxes, for the radius, and press **Enter**.
   d. Type **(0, 0, 242)** in the **(dX, dY, dZ)** boxes, for the height, and press **Enter**.

      The **Segment number** dialog box appears.
   e. Type **24** in the **Number of segments** text box.
   f. Click **OK**.

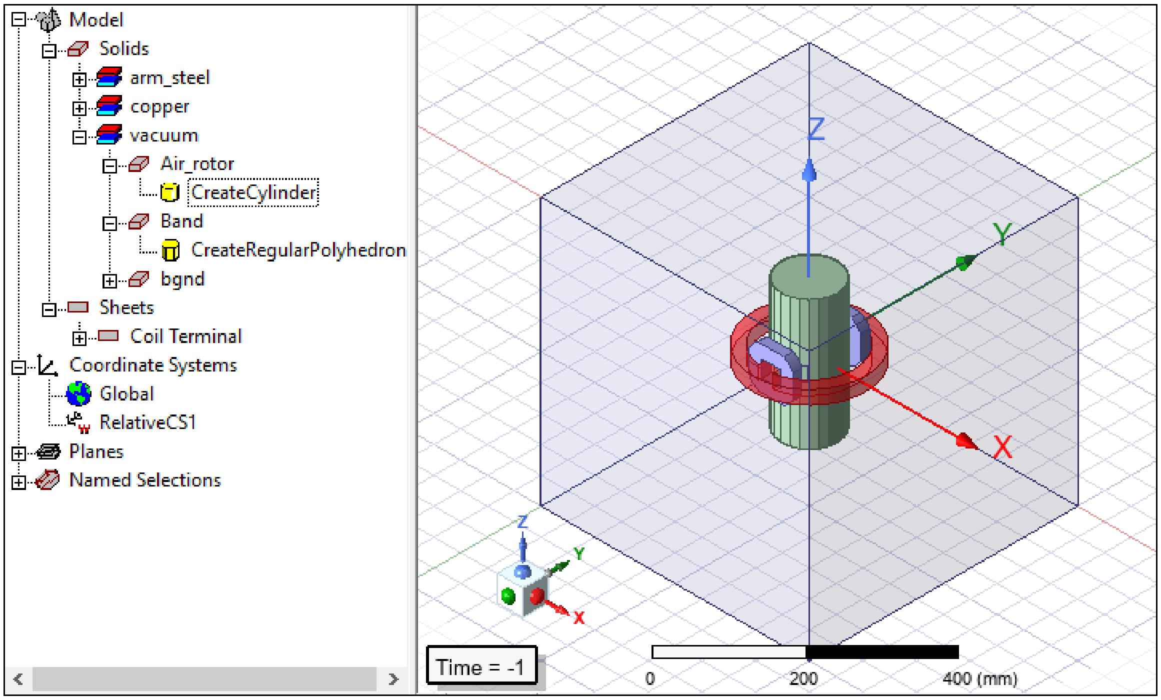
      The **Properties** window appears.
   g. Click the **Attribute** tab.
   h. Change the **Name** to **Band**.
   i. Verify that **Band** is assigned the material property of **vacuum** (which should be the default).

      A polyhedron object named **Band** is drawn.
3. Create a cylinder named **Air_rotor** with the following properties:
   a. Click **Draw** > **Cylinder**.

      The cursor changes to a small black box, indicating that you are in **Drawing** mode.
   b. Enter the center of the cylinder base by typing **(0,0,-120)** in the **(X, Y, Z)** boxes at the bottom of the screen, and press **Enter**.
   c. Type **51.05** for the radius in the **dX** box at the bottom of the screen, and press **Enter**.
   d. Type **240** for the height in the **dZ** box, and press **Enter**.
   e. In the **Properties** window, Click the **Attribute** tab.
   f. Change the **Name** to **Air_rotor**.
   g. Verify that **Air_rotor** is assigned the material property of **vacuum** (which should be the default).

      A cylinder named **Air_rotor** is drawn.

The band and rotor geometry are shown below:



Assign Motion to the Band Object

The circumference of the **Band** object falls between the inner armature and the outer armature and contains inside it the **Air_rotor** and **Inner_arm** objects.

To assign motion to the Band object:

1. Right click on the **Band** from the **Model\Solids** entry in the history tree and select **Assign Band**.

   The **Motion Setup** dialog box appears.

2. From the **Type** tab, select **Rotation** as the **Motion Type**.
3. Set the **Rotation Axis** to **Global:Z** and select the **Negative** radio button.
4. Click the **Mechanical** tab.
5. Select the **Consider Mechanical Transient** check box.
6. Type **0.0024** in the **Moment of Inertia** field.
7. Enter **0.015** in the **Damping** field.
8. Click **OK**.

Apply Meshing to the Band Object

To apply the appropriate mesh operation to the band object:

1. Right click the **Band** object underneath the **Model\Solids** entry in the history tree, and select **Assign Mesh Operation** > **Inside Selection** > **Length Based**.

   The **Element Length Based Refinement** dialog box appears.
2. Enter **20** in the **Set maximum element length** field, and select **mm** as the units.
3. Click **OK**.

Now you are ready to start the analysis with the effect of large motion included.

Set Up the Transient Analysis

To set up a second analysis:

1. Right-click **Analysis** in the **Project Manager** tree, and select **Add Solution Setup**.
2. In the **Solve Setup** dialog box , click the **General** tab.
3. Type **0.9** in the **Stop time** box, and select **s** as the unit.
4. Type **0.005** in the **Time step** box, and select **s** as the unit.
5. Add a sweep:
   a. Click the **Save Fields** tab.
   b. Select **Custom**. A row is added to the table that should have the following values:
      - **Linear Step**. This type-of-sweep setting can also be selected from the Distribution drop-down menu.
      - **0.0** in the **Start** box.
      - **0.9** in the **End** box.
      - **0.005** in the **Step size** box.
6. Click **OK**.

Run the Transient Analysis

To run the analysis:

Under **Analysis** in the Project Manager tree, right-click **Setup2**, and select **Analyze**.

> **Note:** This simulation will take several minutes; the exact simulation time will depend on your computer resources.

Postprocess the Transient Results

For the transient case, additional mechanical quantities are available (to represent as 2D plots as functions of time), which were not available before adding motion to the design. When you create a report for a solution that is set to **Transient**, these new quantities can be added as traces:

- Create a Position vs. Time Plot
- Create a Current vs. Time Plot with Motion
- Create a Torque vs. Time Plot with Motion
- Create a Power Loss vs. Time Plot

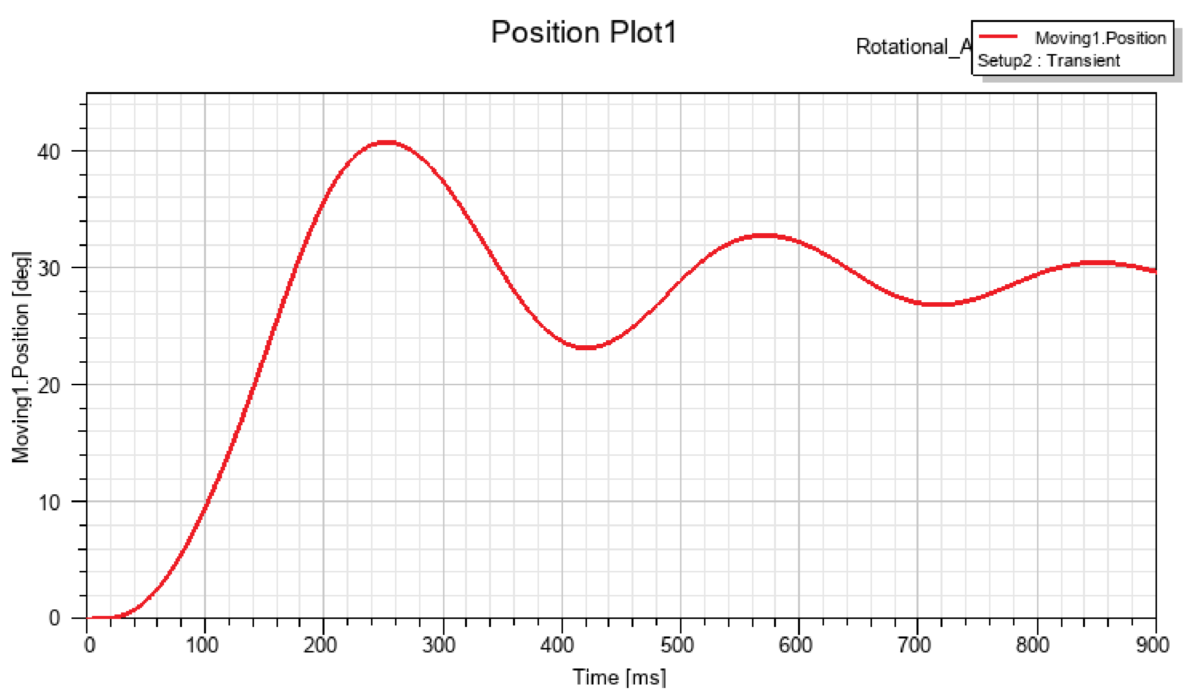
Create a Position vs. Time Plot

To create a plot of the position as a function of time:

1. Right-click **Results** in the Project Manager tree, and select **Create Transient Report** > **Rectangular Plot**.

   The **Traces** dialog box appears.

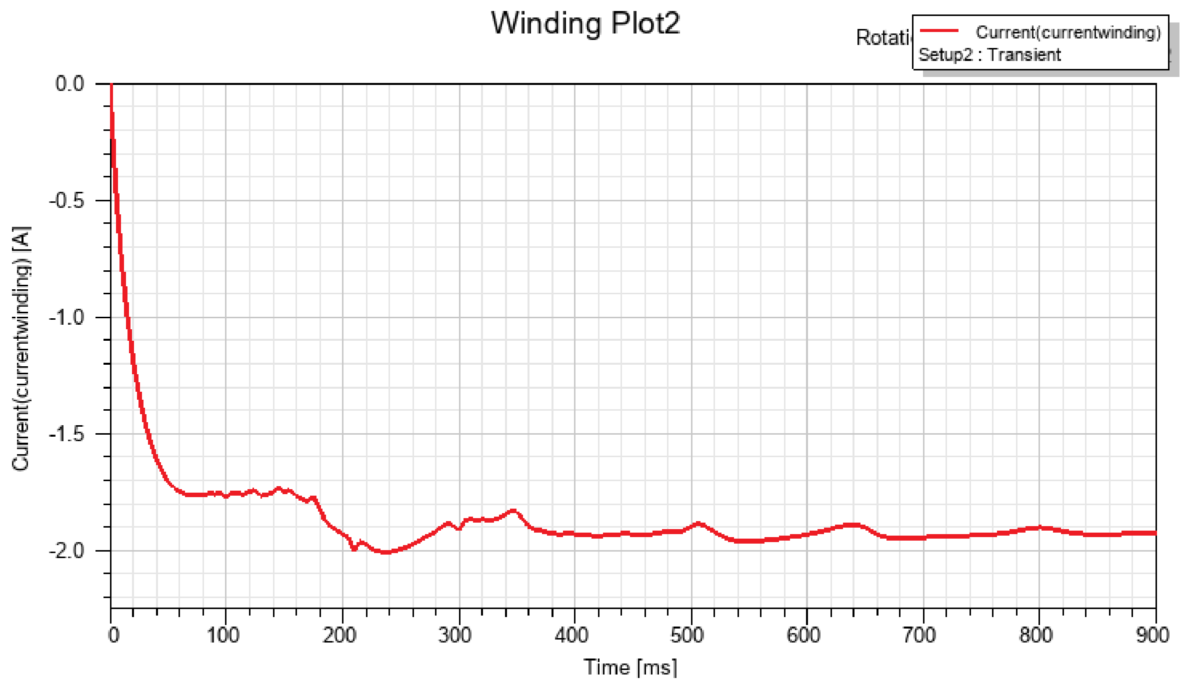
2. From the **Solution** drop-down menu, select the solution setup (**Setup2**).
3. From the **Category** list, select **Position**.
4. Click the **New Report** button.
5. Leave the Traces dialog box open.



Current vs. Time Plot with Motion

To create a plot of the current as a function of time:

1. Right-click **Results** in the Project Manager tree, and select **Create Transient Report** > **Rectangular Plot**.
2. From the **Solution** drop-down menu, select the solution setup (**Setup2**).
3. In the **Traces** dialog box, from the **Category** list, select **Winding**.
4. From the **Quantity** list, select **Current**.
5. Click the **New Report** button.

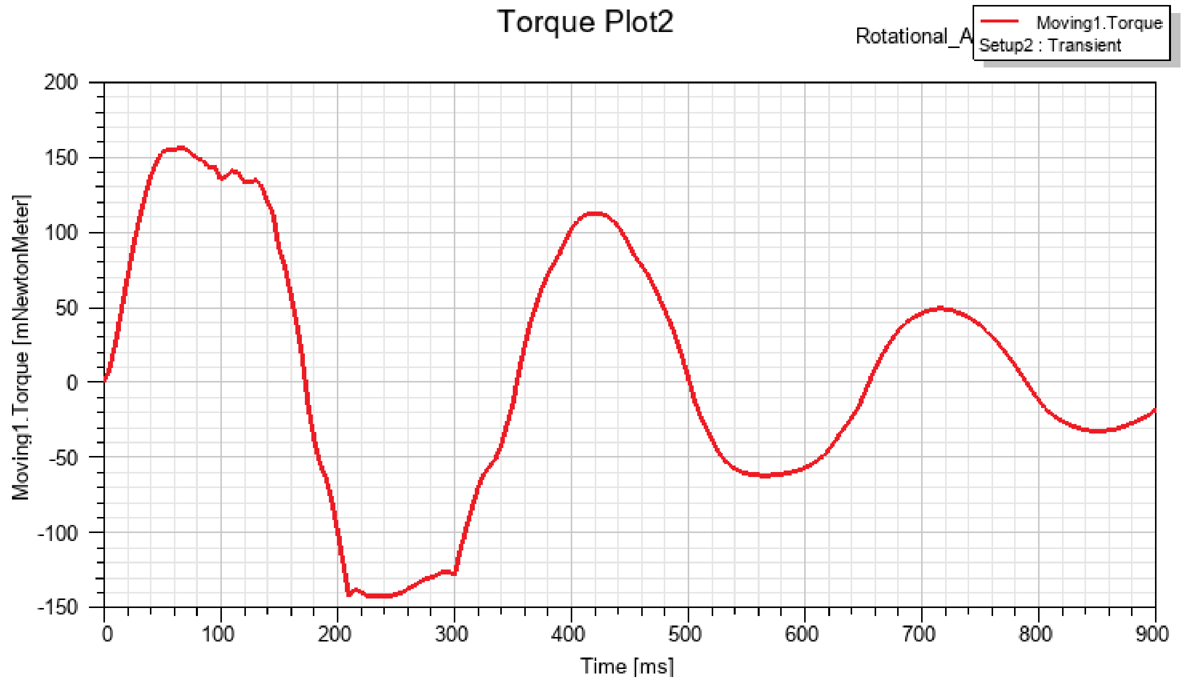


Torque vs. Time Plot with Motion

To create a plot of the torque as a function of time:

1. In the Traces dialog box, from the **Category** list, select **Torque**.
2. From the **Quantity** list, select **Moving1.Torque**.

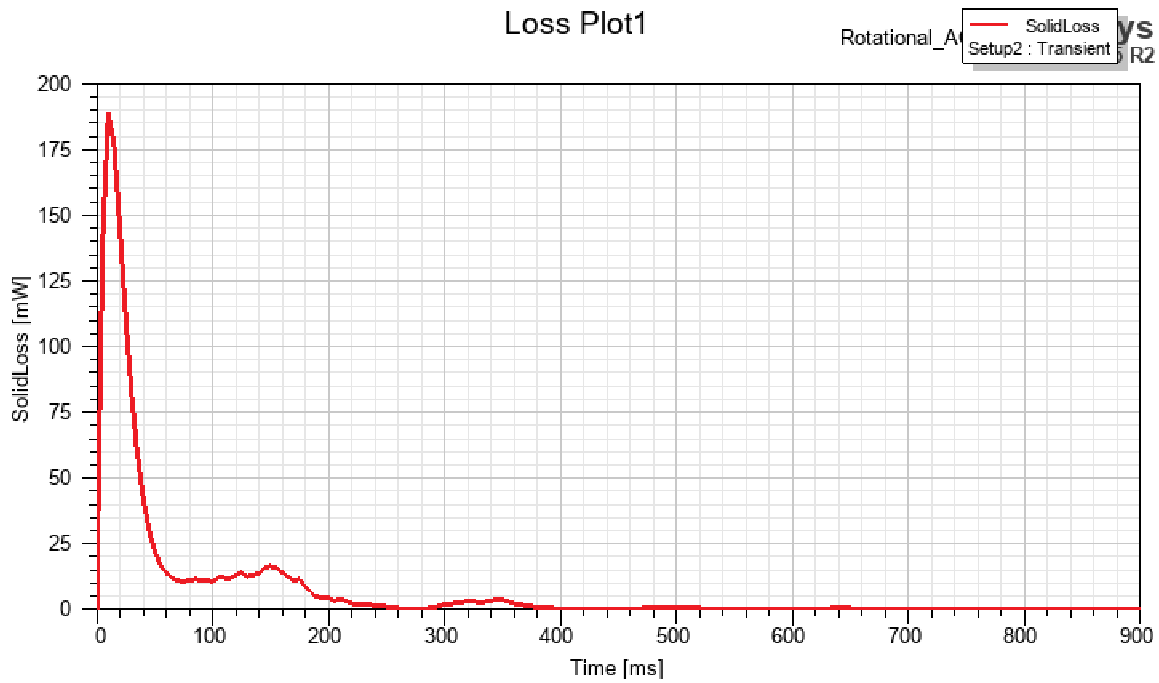
3. Click the **New Report** button.



Create a Power Loss vs. Time Plot

To create a plot of the eddy current power loss in the **Inner_Arm** and **Outer_Arm** as a function of time:

1. In the **Traces** dialog box, from the **Category** list, select **Loss**.
2. From the **Quantity** list, select **Solid Loss**.
3. Click the **New Report** button.
4. Click **Close** to dismiss the dialog box.


Loss Plot1
Rotational_A
SolidLoss
Setup2 : Transient
ys
R2
200
175
150
125
100
75
50
25
0
SolidLoss [mW]
0
100
200
300
400
500
600
700
800
900
Time [ms]

8 - Close the Project and Exit Electronics Desktop

Congratulations! You have successfully completed his *Getting Started with Maxwell: Transient Problem*! You may close the project and exit the Ansys Electronics Desktop software.

1. Click **File** > **Save** to save the project.
2. Click **File** > **Close**.
3. Click **File** > **Exit** to exit Electronics Desktop.