



Ansoft RMxpert Application Note

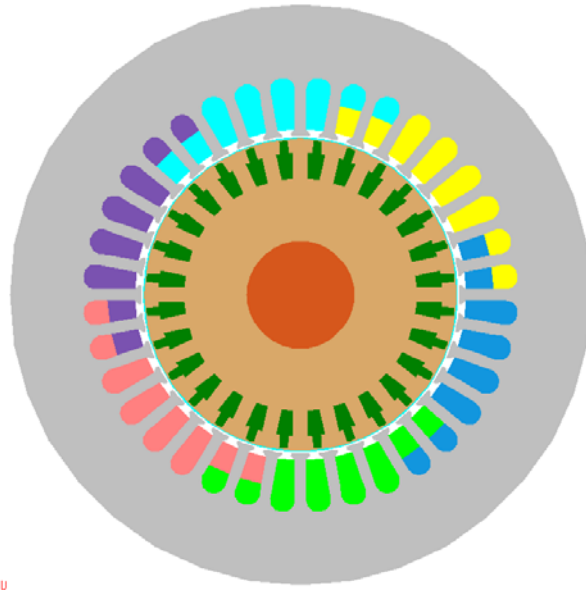
A Three-phase Induction Motor Problem

This application note describes how to set up, solve, and analyze the results of a two-pole, three-phase induction motor using RMxpert. The model created by RMxpert will then be used as an input for EMpulse as a basis for performing more detailed analysis and for increasing the accuracy of the performance predictions.

RMxpert uses a combination of analytical and magnetic circuit equations to predict the performance of this three-phase induction motor. EMpulse is a nonlinear time-domain finite element analysis (FEA) software package that solves electromagnetic field equations, electric circuit equations, and equations of motion.

You can create the project from scratch or open the pre-solved project called **3ph.pjt**, in the **/ansoft/examples/rmxprt/** directory. If you are creating a project from scratch, select **Three Phase Induction Motor** as the motor type in RMxpert.

This project was created using version 2.3 of RMxpert and version 7.0 of the Maxwell 2D Field Simulator.



General Data

Use the **General** window to specify the motor characteristics.

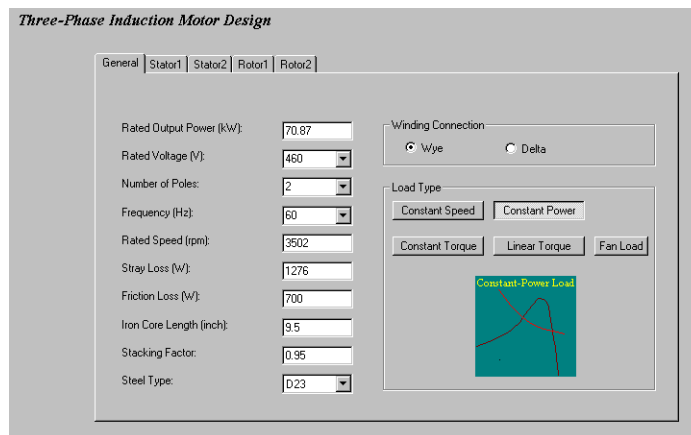
➤ Define the general data:

1. Enter **70.87 kW** in the **Rated Output Power** field for this two -pole motor. This is equal to a 95 horsepower motor.
2. Enter **460 V** in the driving RMS line-to-line **Rated Voltage** field.
3. Enter **60 Hz** in the **Frequency** field.
4. Enter **3502 rpm** in the **Rated Speed** field.
5. Enter **1276 W** in the **Stray Loss** field. If the measured stray load loss is unavailable, NEMA MG1 [1], paragraph 20.52, states that this value shall be assumed to be 1.2% of the rated output for motors rated less than 2500 hp and 0.9% for motors rated 2500 hp and greater. IEEE Standard 112 [2] gives different assumed stray load loss values for motors rated less than 2500 hp. They are as follows:

1-125 hp	1.8%
126-500 hp	1.5%
501-2499 hp	1.2%

In this example, follow the IEEE guidelines and use 1.8% or 1276 watts.

6. Enter **700 W** in the **Friction Loss** field. This value contains both the friction and wind losses.
7. Choose **Tools/Model Units**, and select **Inches** as the units. Choose **OK** to accept the units and close the window.
8. Enter **9.5 inches** in the **Iron Core Length** field to define the length of the stator.
9. Enter **0.95** in the **Stacking Factor** field. This gives a value of 9.025 inches as the net length of the steel, after taking lamination into account.
10. Select **D23** as the nonlinear **Steel Type** used in the manufacturing of the stator lamination. To examine the material BH-curve for D23, choose **Materials/BH**, and then, in the **Material Data** window, choose **Open**, and select **D23.h-b**. Once the data is loaded in the left window, you can plot the B-H curve for this material. The standard loss and specific weight of the material are also displayed. This information is necessary to calculate the iron core loss. Exit this window, and continue the general data input.
11. Select **Wye** as the **Winding Connection**.
12. Select **Constant Power** as the **Load Type**.



Stator Data

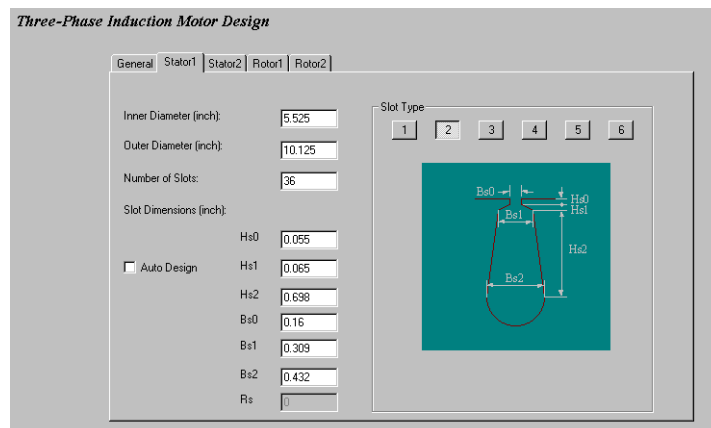
Use the **Stator1** and **Stator2** windows to define the stator dimensions and stator windings.

Define the Dimensions

Use the **Stator1** window to define the stator dimensions.

➤ Define the stator:

1. Enter **5.525** inches in the **Inner Diameter** field.
2. Enter **10.125** inches in the **Outer Diameter** field.
3. Enter **36** in the **Number of Slots** field.
4. Select **2** as the **Slot Type** for the shape of the slot.
5. Deselect **Auto Design** if it is selected, and enter the following **Slot Dimensions** (inch):
 - Enter **0.055** in the **Hr0** field.
 - Enter **0.065** in the **Hs1** field.
 - Enter **0.698** in the **Hs2** field.
 - Enter **0.16** in the **Bs0** field.
 - Enter **0.309** in the **Bs1** field.
 - Enter **0.432** in the **Bs2** field.

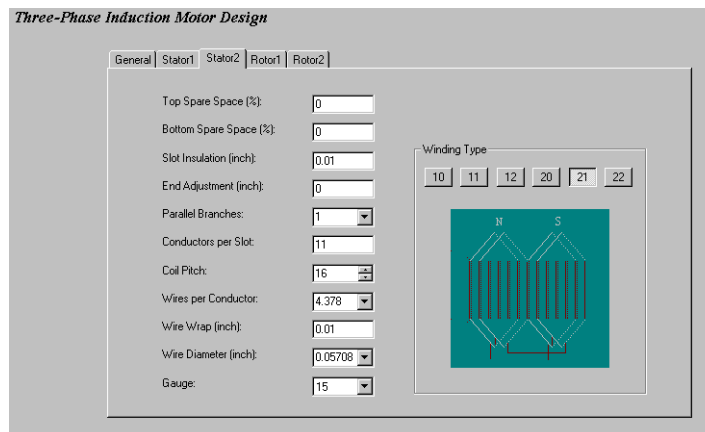


Define the Windings

Use the **Stator2** window to define the stator windings.

► Define the windings:

1. Select **21** in the **Winding Type** field. The winding is a double layer lap winding.
2. Enter **0** in both the **Top Spare Space** and **Bottom Spare Space** fields. The spare spaces in the slot are 0% of the slot's area.
3. Enter **0.01** inches in the **Slot Insulation** field. The slot insulation is the thickness on one side of the stator slot.
4. Enter **0** in the **End Adjustment** field. The end adjustment is the length that the stator windings extend beyond the end of the stator. For this motor, the windings do not extend beyond the stator.
5. Enter **1** in the number of **Parallel Branches** field. This means the coils making up one phase are connected in series.
6. Enter **11** in the number of **Conductors per Slot** field.
7. Enter **16** in the **Coil Pitch** field. Had the stator used a full pitch winding, then the coil pitch would be 18 slots (36 slots/2 poles). The winding is wound 2 slots shorter, which results in a coil pitch of 16 slots.
8. Enter **4.378** in the number of **Wires per Conductor** field. This is not the actual number but the equivalent number that gives the total wire cross-sectional area. (One conductor is made up of two 15 AWG wires and of three 16 AWG wires.)
9. Enter **0.01** inches in the thickness of the **Wire Wrap** field.
10. Enter **0.05708** inches in the **Wire Diameter** field.
11. Enter **15** in the **Gauge** field. Gauge settings are given in AWG.



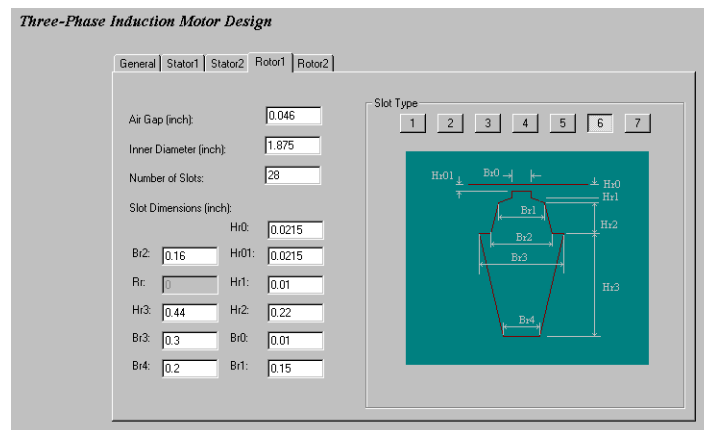
Rotor Data

Use the **Rotor1** and **Rotor2** windows to define the rotor.

Define the Slot Data

Use the **Rotor1** window to define the rotor measurements and the characteristics of the rotor slots.

- Define the rotor measurements and rotor slot characteristics:
 1. Enter **0.046** inches in the **Air Gap** field to define the width of the air gap between the rotor and the stator.
 2. Enter **1.875** inches in the **Inner Diameter** field to define the inner diameter of the rotor. This value is the same as the diameter of the shaft.
 3. Enter **28** in the **Number of Slots** field to define the number of rotor slots.
 4. Select **6** as the **Slot Type**.



Define the Vent Data

Use the **Rotor2** window to define the rotor vent characteristics and other rotor measurements.

- Define the rotor vent characteristics and rotor measurements:
 1. Enter **0** in the **Numbers of Vents** field. There are no vents in the rotor.
 2. Under **Skew Width**, select **Other**, and enter **0** inches as the skew.
 3. Enter **0.815** inches in the **End-Ring Height** field.
 4. Enter **1.276** inches in the **End-Ring Width** field.
 5. Under **Conductor Resistivity**, select **Aluminum** to define the resistivity. This is the material used in manufacturing the bars and the end ring.

Process the Analytical Design

With the motor data defined, you are ready to generate the model.

- Generate the model:
 1. Choose **Tools/Options** to make certain that the **Wire Setting** is **American**, and then choose **OK** to close the window.
 2. Choose **Analysis/Analytical Design**. RMxpert calculates the motor performance parameters for this design.

Displaying the Lamination

Once the analysis is completed, you can display the laminations on the objects.

- Display the laminations:
 1. Choose **Tools/Options/Lamination** and make sure that all the items are checked. Choose **OK** to close this window when you are finished.
 2. Choose **Post Process/View Lamination** to examine the cross-section of the motor. Choose **OK** to close the window when have finished viewing.
 3. Choose **Post Process/View Winding Layout** to see the winding arrangement.
 4. Choose **Exit** when you have finished viewing.

Design Output

Choose **Post/Process/Design Output** to examine the motor's parameters. The **Design Output** window is broken down into the following sections: General Data, Stator Data, Rotor Data, Rated-Load Operation, No-Load Operation, Break-Down Operation, Locked Rotor Operation, Detailed Data at Rated Operation, Winding Arrangement, and Transient FEA Input Data.

General Data

This information is the same as the data you entered in the **General** window.

Stator Data

This information is generally the same as the data you entered in the **Stator1** and **Stator2** windows. If **Auto Design** was selected, RMXprt displays the optimized values for the windings.

Rotor Data

This information is the same as the data you entered in the **Rotor1** and **Rotor2** windows.

Rated-Load Operation

This section displays information about the main performance parameters in the steady state: current, losses, mechanical torque, and input power, as well as the parameters of the one-phase equivalent circuit: resistance and leakage reactance for the stator winding, and magnetizing reactance for the rotor.

No-load, Break-Down, and Locked Rotor Operation

These sections display information about the motor parameters at different operating conditions.

Detailed Data at Rated Operation

The detailed data includes the slot, end-winding, differential, and skewing leakage reactance for the stator winding and the rotor. The sum of these values gives the main leakage reactance as displayed in the Rated-Load Operation section. Other data listed in this area include the windings factor; the flux density for the stator-teeth, rotor-teeth, stator-yoke, rotor-yoke, and air gap; and magnetomotive force:

Slot Fill Factor	The percentage of the available slot area (total slot area minus slot insulation) that is filled with the wire (copper plus insulation).
Correction Factor	The correction factor for the magnetic circuit length for the stator yoke and rotor yoke.
Saturation factor	The saturation factor for the Teeth and the Teeth and Yoke fields.
Stator Current Density	The current density of the stator.
Specific Electric Loading	The ampere-conductors per meter of armature periphery.
Mean Half-Turns Length	The mean number of half-turns for the main and auxiliary phases.
Stator Thermal Load	The current density in each slot multiplied by the Specific Electric Loading.
Half-Turns Length of Stator Winding	The half-turns length of the stator windings.

Winding Arrangement

This is the winding arrangement for one full A phase, B phase, and C phase winding. Only one layer arrangement is displayed; the second can be deduced rapidly from the coil pitch. For this example, the winding arrangement is:

Top layer: AAAAAAZZZZZZBBBBBBXXXXXXXXCCCCCYYYYYY
Bottom layer: YYAAAAZZZZZZBBBBBBXXXXXXXXCCCCCYYYYYYAA

The **Winding Arrangement** section also displays the following values (in electrical degrees):

Angle per slot 180 electrical degrees divided by the number of slots per pole.
Phase-A axis The center of the A phase winding with respect to the phase first slot.
First slot center The reference used to calculate phase.

Transient FEA Input Data

This information is used when calculating the motor performance using the 2D time transient finite element field solver.

For the main and auxiliary windings, this section displays:

- the numbers of turns, as seen from the terminal.
- the number of parallel branches.
- the terminal resistance.
- the end leakage inductance.

For the rotor end ring, this section displays both the end ring resistance and the end ring inductance between two bars at one of the ends.

When you have reviewed the output data, choose **Exit** to exit the window and begin plotting the performance curves.

Plotting the Performance Curves

► Plot the curves:

1. Choose **Post Process/Performance Curves**. The **PlotData** window appears, with an **Open** window visible. The following plot titles are available to open:

n_curr.dat Input Current vs Speed

n_effi.dat Efficiency vs Speed

n_pow2.dat Output Power vs Speed

n_powf.dat Power Factor vs Speed

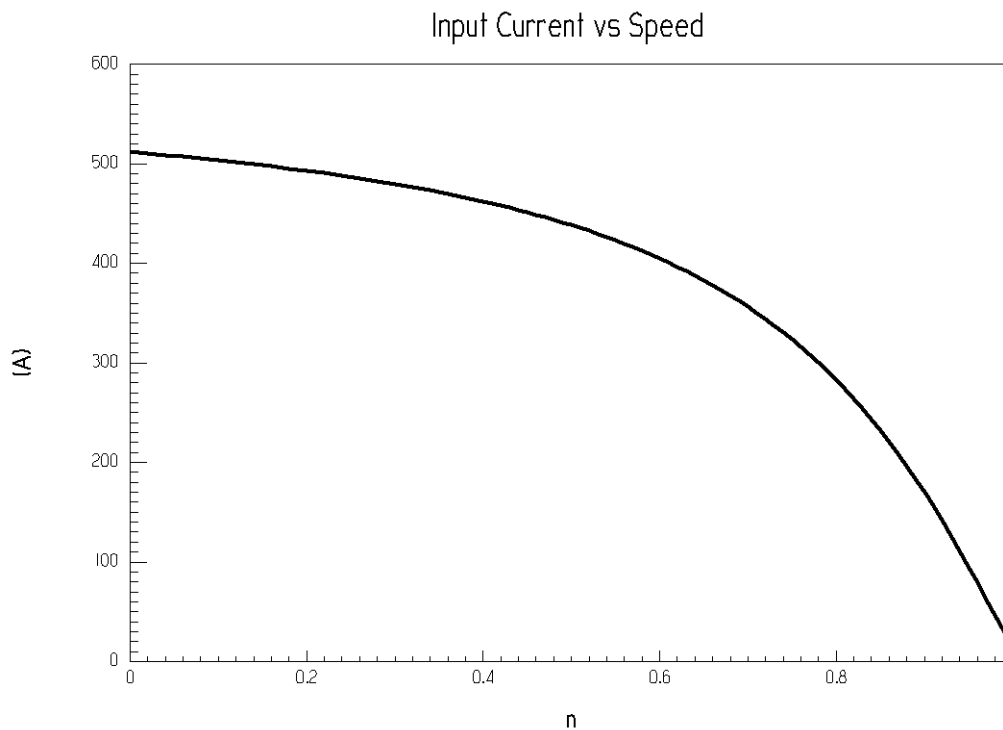
n_torq.dat Output Torque vs Speed

2. Select the name of the plot to view.

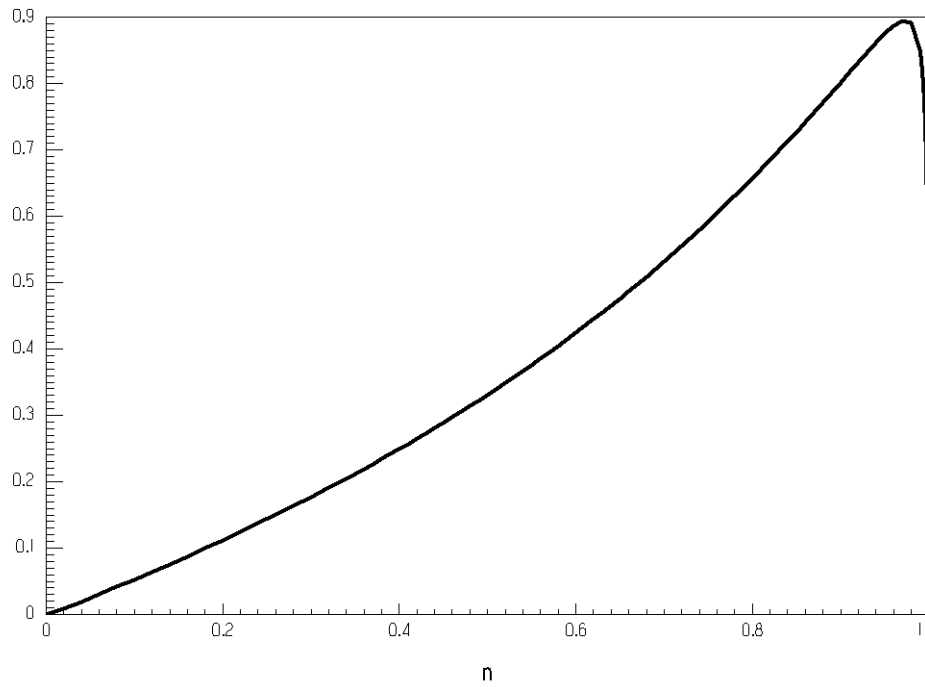
3. Choose **OK**. The plot appears in the **PlotData** window. After you've opened one plot, to open a different plot, choose **Plot/Open**.

The speed is measured per unit of the synchronous speed.

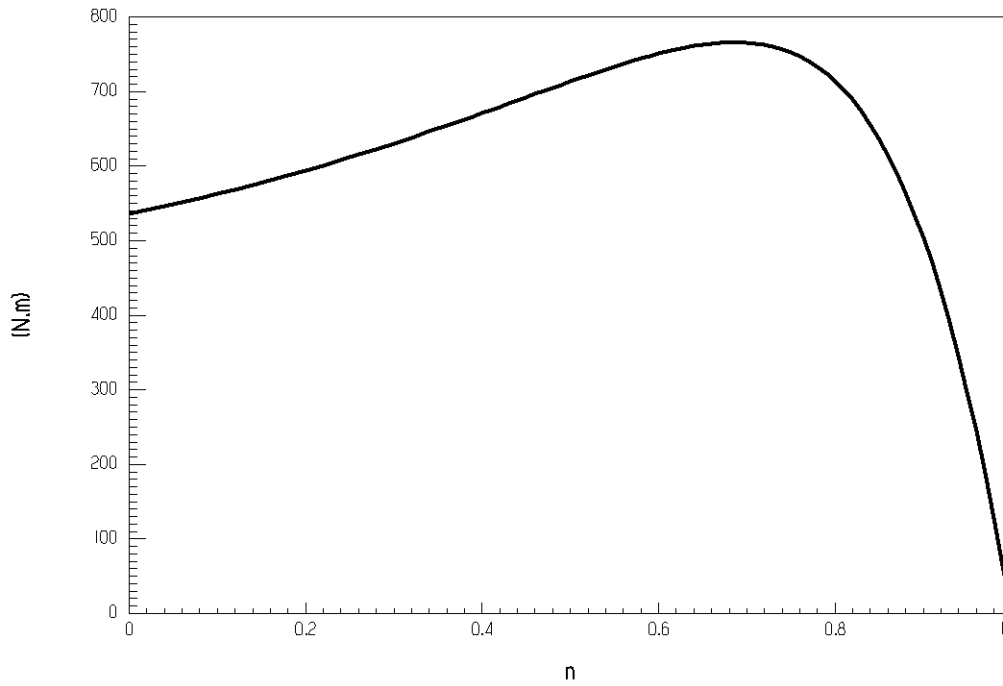
The following figures show the performance plots for the sample problem:



Efficiency vs Speed



Output Torque vs Speed



4. When you have finished viewing the performance curves, choose **File/Exit** to exit PlotData.

Create the Maxwell 2D Project

Now you can create the Maxwell 2D project.

- Create the Maxwell project:
 1. Choose **Tools/Options**, and make certain that the **Periodic** and **Teeth-Teeth** options are selected, and then choose **OK** to close the window. These options allow you to take advantage of the motor symmetry.
 2. Choose **Analysis/View Geometry**. The Maxwell 2D Modeler appears, displaying the geometry.
 3. Choose **File/Exit** to exit the Maxwell 2D Modeler.
 4. Choose **Analysis/Create Maxwell 2D Project**. A message window appears, informing you that the Maxwell 2D project has been created.
 5. Choose **OK** to close the message window.
 6. Return to the Project Manager to continue with the rest of this example. Leave RMXprt open to refer to later in the example.

Exporting the BH-Curve

Once the project has been created, you can export the BH-curve for the magnetic material HP76F to use for the materials in the Maxwell project.

- Export the BH-curve:
 1. Choose **Materials/BH**. The **Material Data** window appears.
 2. Choose **Open**. A file browser appears.
 3. Locate and select the **D23.h-b** file.
 4. Choose **Open**. The data for the material appears.
 5. Choose **Save as** and save the file as a **.bh** file (not an **.h-b** file). This file can be directly used by the 2D transient solver.
 6. Choose **OK** to save the file and close the browser.
 7. Choose **Exit** to close the **Material Data** window.

This file can now be directly used by the 2D transient solver.

Some Exercises

This section provides some useful exercises for analyzing changes in the motor's performance in different situations.

Y - Delta Switching

Create an induction motor designed for a delta armature connection. Examine the starting torque and current for the two cases of the Y connection and the delta connection.

You should obtain the following results:

- For the delta connection, the torque is 535 Nm, and the phase current is 512 A.
- For the Y connection, the torque is 125 Nm, and the phase current is 247 A.

Multiple Parallel Circuits

The windings forming one phase may be connected in series or in parallel, providing multiple parallel circuits (parallel branches). Examine the changes in the winding resistance and the phase current at the rated-load operation. You should obtain the following results:

- For one parallel branch, the resistance is 0.232 ohms, and the phase current is 64 A.
- For two parallel branches, the resistance is 0.058 ohms, and the phase current is 214 A.

Influence of Rotor Resistivity on Locked-Rotor Torque

Change the material for the rotor bar. Use copper instead of aluminum, and examine the changes in the value of the locked-rotor torque. Keep the Y connection and two parallel paths. You should obtain the following results:

- For aluminum, the locked-rotor torque is 753 Nm.
- For copper, the locked-rotor torque is 582 Nm.

Three-Phase Induction Motor Simulation Using EMPulse

Now that the Maxwell 2D project is created, use the EMPulse environment to predict the operating performance of the motor. EMPulse is a nonlinear time-domain FEA software package that analyzes the electromagnetic (plane parallel) phenomena in electromagnetic devices, combined with electric circuits and motion.

If you want to simply open and inspect the finished project, the pre-solved project is included in **3ph_fea.pjt**. If you want to step through the project yourself, open the project you previously exported from RMxpert.

Set Up the Geometry

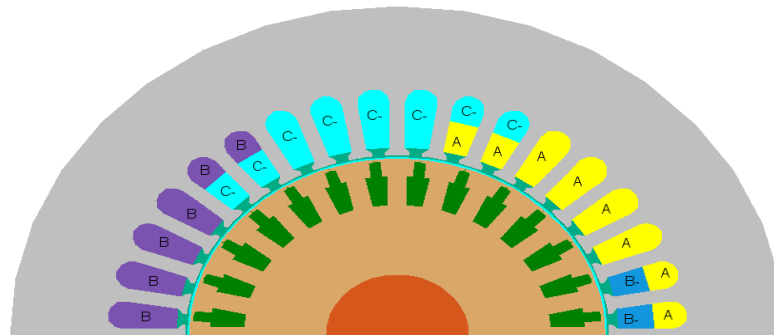
The geometry of the model is already created by RMxpert.

- Open the project, and set up the geometry:
 1. From the Project Manager in the Maxwell Control Panel, open the Maxwell 2D project that was created in the previous section. If using the pre-solved project, its name is **3ph_fea.pjt**. Upon opening the project, notice that the **Transient** solver, the **XY** drawing plane, and **Define Model** are already set.
 2. Choose **Define Model/Draw Model** to open the Maxwell 2D Modeler.
 3. Choose **View/Zoom In**, and zoom in on the air gap. There is an additional object in the air gap called **Band**. This object is used during the solution process to determine which objects are stationary and which objects rotate.
 4. Choose **File/Save**, and then **File/Exit**, to save the file and exit the Maxwell 2D Modeler.
 5. Choose **Define Model/Group Objects**, and group the objects that belong to the same winding. RMxpert assigns names to all of the objects in the geometry. Group the objects as follows (notice the abbreviations for the names of the windings):

Bar	objects Bar0-Bar13
PhA	objects PhA0-PhA11
PhB	objects PhB0-PhB11
PhReB	objects PhReB10, PhReB11
PhReC	objects PhRe0-PhRe11

Note that the stator windings of an induction motor can be grouped into six different objects. Since the FEA model is only half of the full model, groups **PhReA** and **PhC** are not in the sample drawing.

If you assume that the rotor rotates counterclockwise (positive torque), the stator windings have to be grouped counterclockwise in the same sequence. In this example, that order is **PhA** (represented as A), **PhReC** (represented as C-), **PhB** (represented as B), and **PhReB** (represented as B-).



Setup Materials

After grouping the objects, you can now assign material properties. Because this example requires materials not included in the material database, you must create them in the Material Manager.

Choose **Setup Materials** to access the Material Manager and assign material properties to all of the objects.

Add a New Material

Add a new nonlinear material called D23 to the local material database.

- > Add a new material:
 1. Choose **Material/Add**, and enter **D23** in the name field below **Material Attributes**.
 2. Select **Nonlinear Material**, and choose **B H Curve**. The **B-H Curve Entry** window appears.
 3. Choose **Import** from the **The B-H Curve Entry** window. The **Import File** window appears.
 4. Select the **D23.bh** file that was created within RMXprt. Make certain that the **bh** button is selected before importing. In most cases, this file is located in **c:\ansoft\rmxpri\matlib**.
 5. Choose **OK** to import the file and return to the **B-H Curve Entry** window.
 6. Choose **Exit** to exit the window and return to the Material Manager. The new material is now available in the database.

Derive a New Material

Derive a new conductor material called Aluminium_115.

- > Derive the new material:
 1. Select **aluminum** from the **Materials** list.
 2. Choose **Material/Derive**.
 3. Enter **Aluminum_115** in the name field below **Material Attributes**.
 4. Enter **2.304e7 S/m** in the **Conductivity** field.
 5. Choose **Enter** to enter the new material in the local material database.

Assign the Materials

Now assign material properties as follows.

- > Assign the materials:
 - Assign **vacuum** to the **AirGap** and **Band**.
 - Assign **Aluminum_115** to the **Bar** group.
 - Assign **copper** to groups **PhA**, **PhB**, **PhReB**, **PhReC**.
 - Assign **D23** to the **Rotor** and **Stator**.
 - Assign **steel_stainless** to the **Shaft**.
 - Exclude the background from the model.

Note that the rotor bar conductivity is set up for a working temperature of 75 degrees Celsius. The conductivity of the stator windings will not be taken into account if the windings are set up as a stranded conductor. The resistances of the stator windings will be specified in the Source Setup.

Setup Boundary / Sources

The first step in defining the boundary conditions is to define the Master/Slave boundary. You then need to setup the source and assign the end ring parameters.

First, choose **Setup Boundaries/Sources**. The **2D Boundary/Source Manager** appears.

Define the Master Boundary

- Define the master boundary:
 1. Choose **Window/New** to open an additional window. Once the window is open choose **Window/Tile** to arrange the windows in a tile format.
 2. Choose **Edit/Select/Edge**, and select the curved edge of the outside diameter of the stator.
 3. Choose **Assign/Boundary/Value**, and assign to it a value boundary of **0**.
 4. Choose **View/Zoom In**, and zoom in on the air gap so that the area where the **Band** and the inside diameter of the stator cross the x-axis can easily be seen.
 5. Choose **Edit/Select/Trace**. Starting in the window with the full model shown, click on the center axis of the motor ($u=0, v=0$), and then click on the following intersection:
 - Rotor Inside Diameter ($u=23.8125, v=0$)
 6. Switch to the window where the air gap is enlarged, and click on the following intersections:
 - Rotor Outside Diameter ($u=68.9991, v=0$)
 - Band ($u=69.5833, v=0$)
 - Stator Inside Diameter ($u=70.1675, v=0$)
 7. Switch back to the window with the full model, and double-click on the Stator Outside Diameter ($u=128.5875, v=0$) to end the master boundary definition.
 8. Choose **Assign/Boundary/Master**, and then choose **Assign**.

The master boundary is now assigned.

Define the Slave Boundary

Again, use the **Edit/Select/Trace** command to define the slave boundary.

- Define the slave boundary:
 1. Choose **Edit/Select/Trace**. Starting in the window with the full model shown, click on the center axis of the motor ($u=0, v=0$), and then click on the following intersection:
 - Rotor Inside Diameter ($u=0, v=23.8125$)
 2. Switch to the window where the air gap is enlarged, and click on the following intersections:
 - Rotor Outside Diameter ($u=0, v=68.9991$)
 - Band ($u=0, v=69.5833$)
 - Stator Inside Diameter ($u=0, v=70.1675$)
 3. Switch back to the window with the full model, and double-click on the Stator Outside Diameter ($u=0, v=128.5875$), to end the slave boundary definition.
 4. Choose **Assign/Boundary/Slave**, and select **Slave = -Master**. When solving for an odd number of poles of an electrical machine, use the **Slave = -Master** symmetry. When solving for an even number of poles, use the **Slave = +Master** symmetry.
 5. Choose **Assign**.

Source Setup

When specifying the source, you need to consider two options in EMPulse: coils can be either current supplied or voltage supplied. In the electric motors, windings are generally voltage supplied, with the resulting currents dependent on the winding's resistance and back electromotive force (emf). EMPulse can calculate the currents from the field solution and the terminal data.

Phase A Winding

- To set up the Phase A winding parameters:
 1. Choose **Edit/Select/Object/By Clicking**, and then select the object group **PhA**. Click the right mouse button to stop selecting.
 2. Choose **Assign/Source/Solid**. New fields appear below the view window.
 3. Select **Voltage** and **Strand**.
 4. Choose **Functions**, and add the following three new functions:
 - $\text{PhaseA} = 460 \cdot \sqrt{2/3} \cdot \cos(360 \cdot 60 \cdot T)$
 - $\text{PhaseB} = 460 \cdot \sqrt{2/3} \cdot \cos(360 \cdot 60 \cdot T - 120)$
 - $\text{PhaseC} = 460 \cdot \sqrt{2/3} \cdot \cos(360 \cdot 60 \cdot T - 240)$
 5. When you have finished entering the functions, choose **OK** to accept the functions and close the window.
 6. Choose **Options** and select **Function** in the window that appears. Choose **OK** to accept a functional value for the voltage and close the window.
 7. Enter **PhaseA** in the value field.
 8. Choose **Winding**. The **Winding Setup for Boundary** window appears.
 9. Assign a positive polarity to the object group **PhA**.
 10. Under **Terminal Attributes**, enter the following values, which were derived using the transient FEA input data from RMXprt Design Output:
 - Enter **0.231977** ohms in the **Resistance** field. The resistance is the total resistance of the winding at the working temperature
 - Enter **0.000441694** henries in the **Inductance** field. The inductance is only the end-winding leakage inductance, which can not be deduced from the 2D field solution.
 - Enter **66** in the **Total turns as seen from terminal** field.
 - Enter **1** in the **Number of Parallel Branches** field.
 11. Choose **OK** to accept the settings and return to the 2D Boundary/Source Manager.
 12. Enter **PhaseA** in the **Name** field, and then choose **Assign**.

The PhaseA winding parameters are now defined.

Phase B Winding

- To set up the Phase B winding parameters:
 1. Choose **Edit/Select/Object/By Clicking**, and then select the object groups **PhB** and **PhReB**.
 2. Choose **Assign/Source/Solid**. New fields appear below the view window.
 3. Select **Voltage** and **Strand**.
 4. Choose **Options**, and select **Function** for the voltage. Choose **OK**.
 5. Enter **Voltage** in the **Value** field.
 6. Choose **Winding**. The **Winding Setup for Boundary** window appears.
 7. Assign a positive polarity for **PhB** and a negative polarity for the group **PhReB**.
 8. Under **Terminal Attributes**, define the following values:
 - Enter **0.231977** ohms in the **Resistance** field. The resistance is the total resistance of the winding at the working temperature.
 - Enter **0.000441694** henries in the **Inductance** field. The inductance is only the end-winding leakage inductance, which can not be deduced from the 2D field solution.
 - Enter **66** in the **Total turns as seen from terminal** field.
 - Enter **1** in the **Number of Parallel Branches** field.
 9. Choose **OK** to accept the settings and return to the 2D Boundary/Source Manager.
 10. Enter **PhaseB** in the **Name** field, and then choose **Assign**.

Phase C Winding

- To set up the Phase C winding parameters:
 1. Choose **Edit/Select/Object/By Clicking**, and then select object group **PhReC**.
 2. Choose **Assign/Source/Solid**. New fields appear below the view window.
 3. Select **Voltage** and **Strand**.
 4. Choose **Options**, and select **Function** for the voltage. Choose **OK**.
 5. Enter **Voltage** in the **Value** field.
 6. Choose **Winding**. The **Winding Setup for Boundary** window appears.
 7. Assign a negative polarity for group **PhReC**.
 8. Under **Terminal Attributes**, define the following values:
 - Enter **0.231977** ohms in the **Resistance** field. The resistance is the total resistance of the winding at the working temperature
 - Enter **0.000441694** henries in the **Inductance** field. The inductance is only the end-winding leakage inductance, which can not be deduced from the 2D field solution.
 - Enter **66** in the **Total turns as seen from terminal** field.
 - Enter **1** in the **Number of Parallel Branches** field.
 9. Choose **OK** to accept the settings and return to the 2D Boundary/Source Manager.
 10. Enter **PhaseC** in the **Name** field, and then choose **Assign**.

End Ring Parameters

- Define the end connection:
 1. Use the **Edit/Select** commands, and select the object group **Bar**.
 2. Choose **Assign/End Connection**. New fields appear below the view window.
 3. Select the **Passive end-connected conductor** box.
 4. Enter **7.5856e-7** ohms in the **End Resistance between adjacent conductors** field.
 5. Enter **1.47364e-9** henries in the **End Inductance between adjacent conductors** field.

Setup Solution

Since adaptive refinement is not available for the transient solver, the quality of the manual mesh is critical to the accuracy and the convergence of the field solution. The mesh must be fine in regions where a large magnetic field gradient occurs (such as air-gaps and rotor bars) and larger elsewhere. For practical use, generating a mesh that is too fine can result in excessive computational time.

Manual Mesh

- Manually create the mesh:
 1. Select **Setup Solution/Options** from the Executive Commands menu. The **Solve Setup** window appears.
 2. Choose **Manual Mesh**. The 2D Meshmaker appears.
 3. Choose **Mesh/Seed/QuadTree**. The **QuadTree Seed** window appears. Accept **6** as the **Number of levels**, and choose **OK**.
 4. Choose **Mesh/Make**. The basic mesh is generated for the model. This mesh is too coarse to provide the most accurate solution and must be refined. To refine the mesh, you need to take into account those areas which are critical for solution accuracy. In this example, the critical areas are the band, the air-gap, and the rotor bars. During the manual refinement, you can specify the desired number of triangles in each object.
 5. Choose **Refine/Object**. The **Object Refinement** window appears, allowing you to refine the mesh further. The goal is to have a uniform mesh with a sufficient number of elements in the air gap:.
 6. Make certain the following number of triangles appear in the **Refine Number** field for each object:

AirGap	2000
Band	1000
Bars	50
PhA, PhB	40
PhReB, PhReC	40
Rotor	1300
Shaft	150
Stator	2000

If the number of elements for an object does not fall within a few percent of that listed, select its name from the **Object Name** list, and enter its appropriate value in the **Refine Number** field. Choose **Accept** to accept the new value. Values for the air gap, band, and stator require the greatest number of elements to obtain the most accurate solution.

7. Choose **OK** when finished refining the objects.

Refine the Mesh

You can modify the mesh still further by adding individual points with the mouse. In this step, pay particular attention to the region surrounding the air gap and the top of the rotor bars because this region will experience the highest rate of error, requiring the most refinement.

- Refine the mesh:
 1. Choose **Refine/Point**. The **Point Refinement** window appears.
 2. Leave **Circumcircle** selected and choose **OK** to accept this style of point refinement. The window closes.
 3. Add triangles where they are needed to further refine the mesh, if necessary. Click the right mouse button to exit.
 4. Choose **Mesh/Line Match** and select the edges of both the master and slave boundaries, to ensure that the meshing points will match at your matching boundaries. Click the right mouse button when done selecting. If the meshing points do not match at matching boundaries, you will receive an error message about a missing transcript file during the nominal solution.
 5. Choose **File/Exit**, and save the changes to the mesh as you exit from the window.

When you return to the **Solve Setup** window, the **Starting Mesh** option is changed to use the **Current** mesh.

Solution Options for the Transient Analysis

For this example, use the following settings to define the solution options for the transient analysis.

- Define the solution options:
 1. Leave **Starting Mesh** set to **Current**.
 2. Accept the default value in the **Solver Residual** field. This value has no effect when the direct solver is used.
 3. Select **Direct** as the **Solver Choice**. Use this option whenever you generate a solution using the transient solver.
 4. Select **Start from time zero** as the **Solution** starting point.
 5. Enter **0.10** seconds in the **Stop time** field. You will only generate a solution for the first 0.1 seconds of motion.
 6. Enter **0.00015** in the **Time step** field. This instructs the solver to calculate the fields every 150 microseconds of the solution process. For fixed time steps, such as these, typically use 50 steps per electrical cycle. The resulting rotor displacement per one time step should be a maximum of 3 mechanical degrees.
 7. Enter **0.05** in the **Save fields time step** field. This instructs the solver to write the field solution out every 50 milliseconds.
 8. Enter **241.3** mm in the **Model depth** field.
 9. Enter **2** in the **Symmetry multiplier** field. Because you are modeling only one-half of the model, use this multiplier to generate a solution for the entire geometry.
 10. Choose **OK** to accept the values and return to the Executive Commands menu.

Motion Setup

With the solution parameters defined, you can now define the motion parameters for the transient model.

- Define the motion setup:
 1. Choose **Setup Solution/Motion Setup** from the Executive Commands menu. The **Motion Setup** window appears.
 2. Select the **Band** object, and then choose **Set Band**. The band is defined as a stationary object that contains all moving objects.
 3. Select **Rotation** as the **Type of Motion**.
 4. Choose **Set Position**, and select (0,0) as the center of rotation.
 5. Choose **Mechanical Setup**. The **Mechanical Setup** window appears.
 6. Deselect **Consider Mechanical Transient**, and enter **3502** in the **Constant Angular Velocity** field. Make sure the units list beside the field is set to **rpm**.
 7. Choose **OK** to close the window.
 8. Choose **Exit** to exit the **Mechanical Setup** window. Save the changes as you exit.

Solve the Nominal Problem

Choose **Solve/Nominal Problem** from the Executive Commands menu. The progress bar informs you of the status of the solution. Solving the problem takes about 8 hours on a 400 MHz PC.

If you want to display the transient data, such as voltages, currents, torque, and power loss, choose **Solutions/Transient Data**. If you choose **Refresh** during the solution process, the plots will be refreshed after completing the current time step.

Post Processing

➤ Access the Fields Post Processor:

1. Choose **Post Process/Fields**. The **Post-Process Saved Fields** window appears, listing the time steps.
2. Select the time step to plot, and choose **OK**. The 2D Post Processor opens.

Calculate the Flux Density

Calculate the flux density in the air gap. First, you must identify a line in the air gap, then load the B vector and calculate the magnitude of B, map the magnitude of B onto this line, and then take its average value.

➤ Calculate the flux density:

1. Choose **Post/Line/Define**. The **Line Segment Menu** window appears.
2. Zoom into the air gap so that the band can be selected.
3. Enter **Object** in the **Line segment name** field.
4. Select the band object.
5. Choose **Display** and then **Return**.
6. Choose **Calc/Plane** to enter the plane calculator.
7. Under **Transient**, choose **B_Vector** to load the B vector into the calculator.
8. Under **Vector**, choose **magnitude** to calculate the magnitude of B.
9. Choose **Line** to the right of the calculator to access the line calculator.
10. Under **Register**, choose **enter**. Set **lineseg1** to **Yes**, and choose **Execute**.
11. Under **Operations**, choose **value**, and enter **1000** in the blank field. This maps the value of the magnitude of B onto the line.
12. Under **Operations**, choose **integrate**. Set **Enter in number calculator** to **Yes**.
13. Choose **Plane** to return to the plane calculator.
14. Under **Scalar**, choose **constant**, and enter **1.0** in the field.
15. Choose **Line** to return to the line calculator.
16. Under **Register**, choose **enter**. Set **lineseg1** to **Yes**, and choose **Execute**.
17. Under **Operations**, choose **value**.
18. Under **Operations**, choose **integrate**, and set **Enter in number calculator** to **Yes**. This calculates the length of the line.
19. Choose **Number** to enter the number calculator.
20. Under **Operations**, choose **divide** to divide out the length of the line.

Following is the equation we just solved:

$$B(avg) = \frac{\int B \cdot dl}{\int dl}$$

The value in the top register of the calculator stack is the value in tesla of the average flux density in the air gap, which should be approximately 0.412 tesla.

The value calculated by RMxpert is 0.447 tesla.

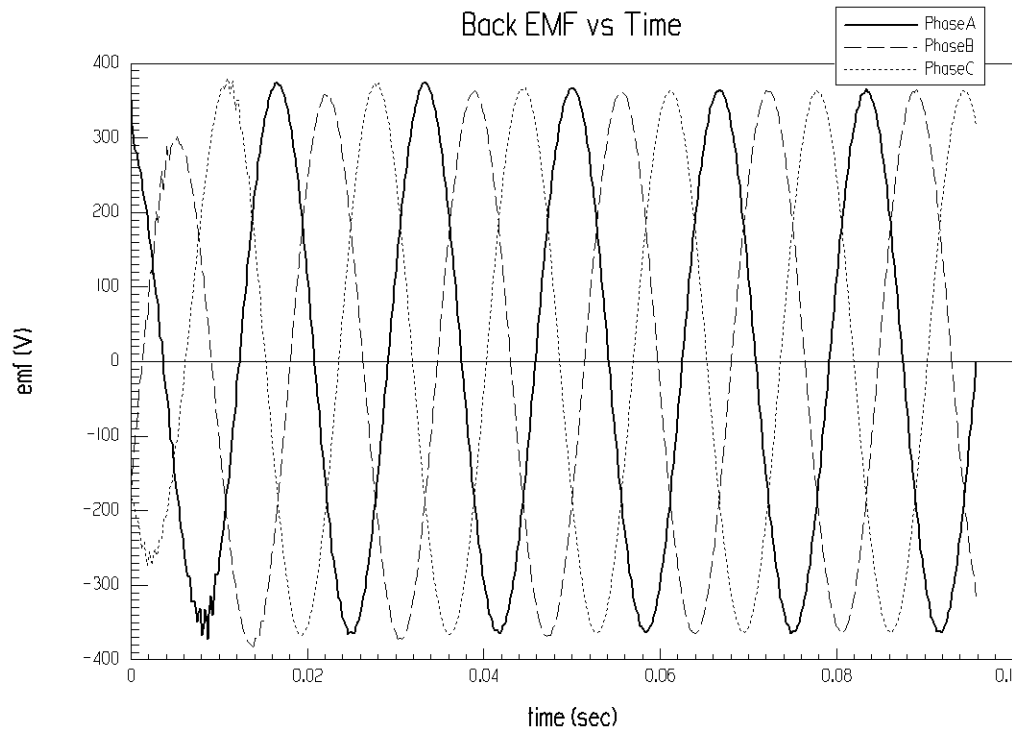
Exit from the calculator, and choose **Post/Plot** to view field quantities such as flux, B, and H.

Results

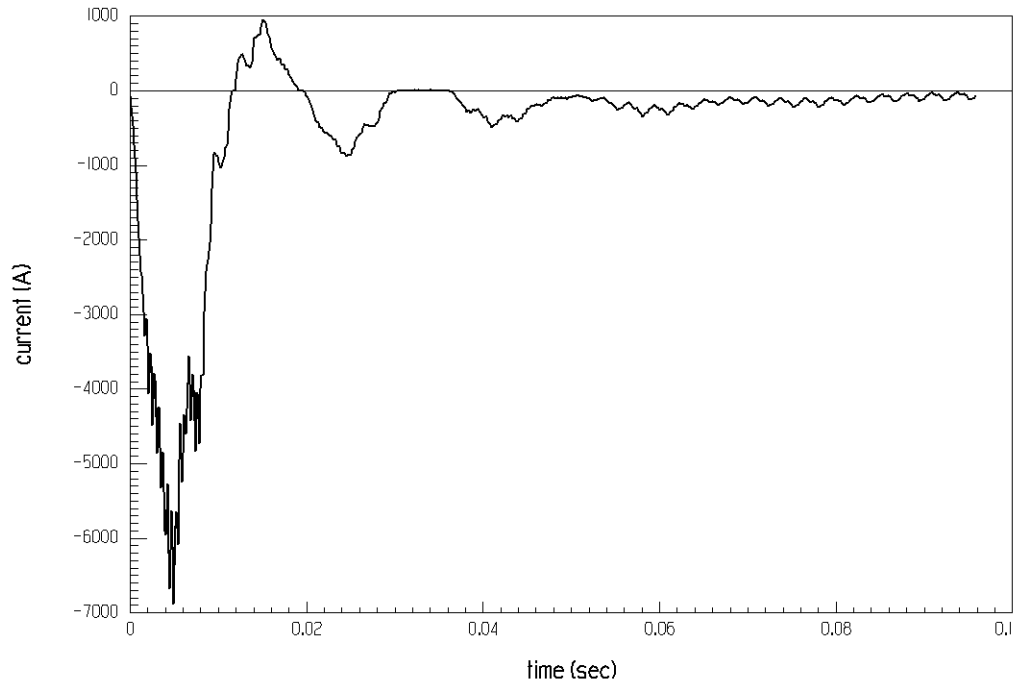
Your values may differ slightly from these, but should still be close:

- Phase Current = 30 A rms (compared to 31 A rms in RMxpert)
- Torque = 58.5 Nm (compared to 56.6 in RMxpert)
- Power Loss = 2230 W

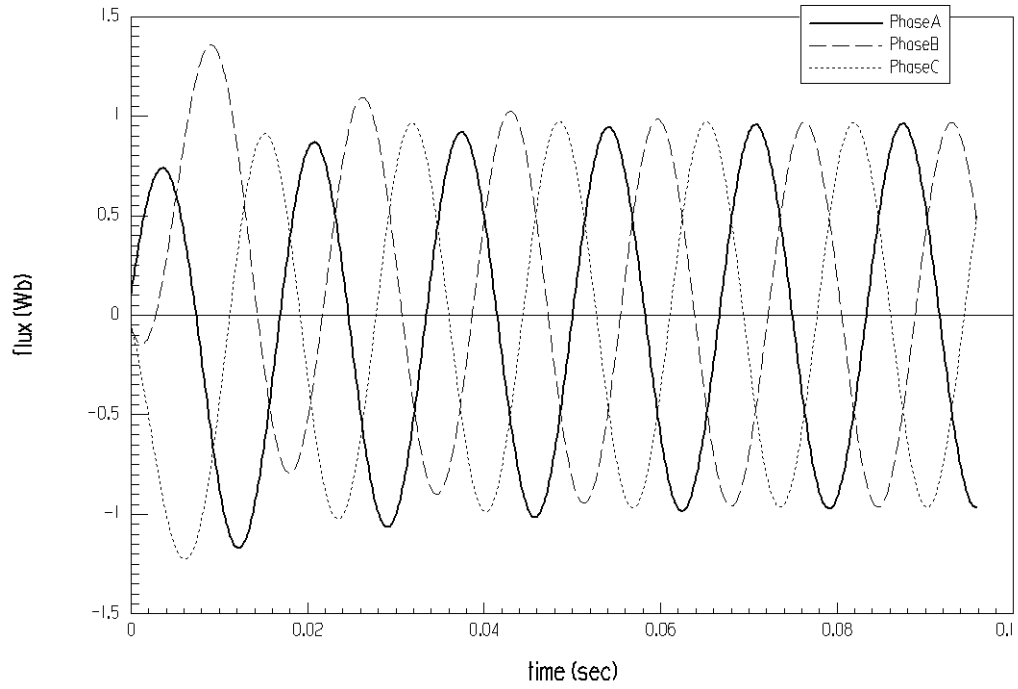
The following figures show the transient plots for the sample problem:

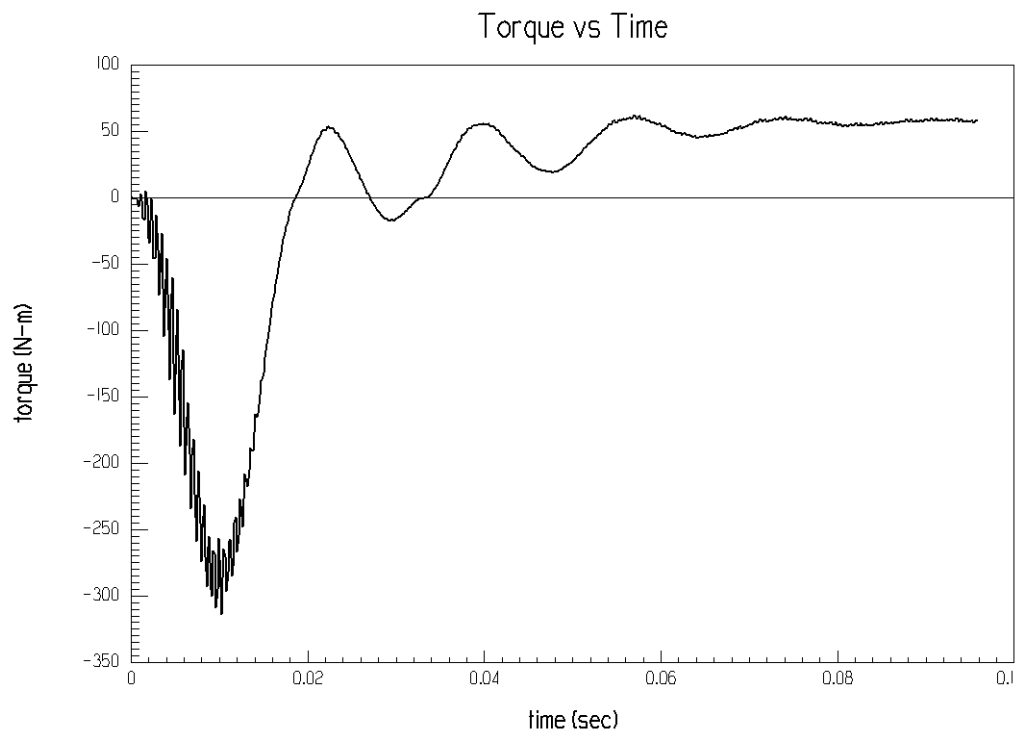


Conductor current vs time



Flux Linkage vs Time





References

[1] NEMA Standards Publication NO. MG1, National Electrical Manufacturers Association (NEMA), Washington D.C.,1993.

[2] IEEE Standard Test Procedure for Polyphase Induction Motors and Generators, IEEE Standard 112-1991, 1991.

