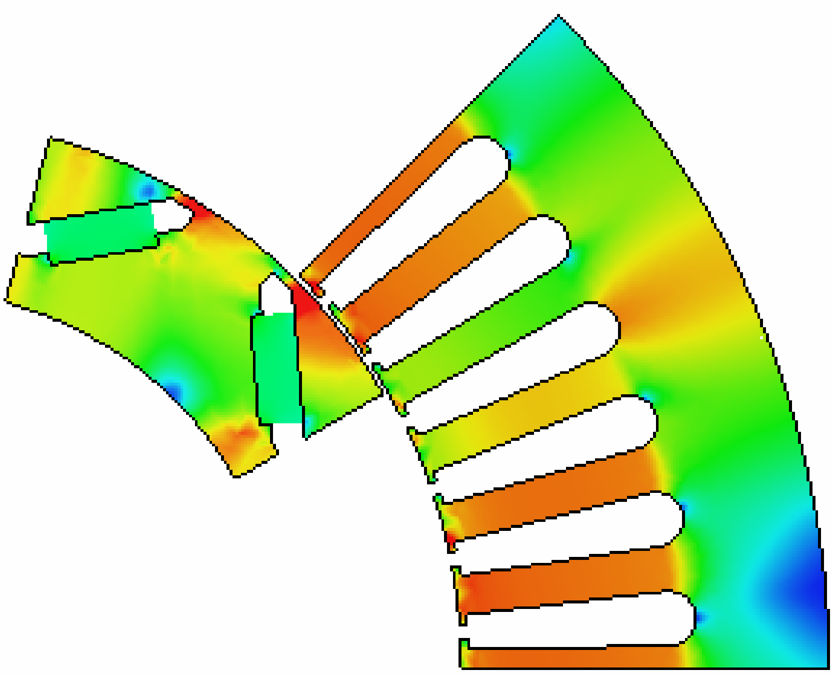
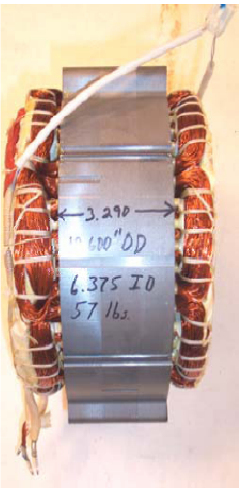
Study of a Permanent Magnet Motor with MAXWELL 2D: Example of the 2004 Prius IPM Motor



Study of an electrical machine

- The Electro Mechanical software package provided by Ansoft enables extensive electrical malchinesimulation. This application note details the simulaton of an electrical machine with Maxwel2D. We will cover static and transient simulations.
- This application note will use the 2004 Toyota Prius motor as basis. It is a 8-pole permanent magnet motor with embedded magnets. The single layer windings are made of 3 phases. The stator has 48 slots. This motor is public, we therefore have the full set of parameters. We will also use Oak Ridge National Laboratory testing results in this note.

> Note: This application has not been done with the collaboration of Toyota



- References:
  - **Report on Toyota/Prius Motor Torque Capability, Torque Property, No-Load Back EMF, and Mechanical Losses,**
    - J. S. Hsu, Ph.D., C. W. Ayers, C. L. Coomer, R. H. Wiles
    - Oak Ridge National Laboratory
  - **Report on Toyota/Prius Motor Design and manufacturing Assessment**
    - J. S. Hsu, C. W. Ayers, C. L. Coomer
    - Oak Ridge National Laboratory
  - **Evaluation of 2004 Toyota Prius Hybrid Electric Drive System Interim Report**
    - C. W. Ayers, J. S. Hsu, L. D. Marlino, C. W. Miller,G. W. Ott, Jr.,C. B. Oland
    - Oak Ridge National Laboratory

Overview of the Study:

- GETTING STARTED
  - Creating the 3D Model
  - Reducing the size of the 3D Model
  - Material properties of the machine
  - Applying Master/Slave Boundary Condition
- STATIC ANALYSIS
- DYNAMIC ANALYSIS
- COGGING TORQUE

Getting Started

Launching Maxwell

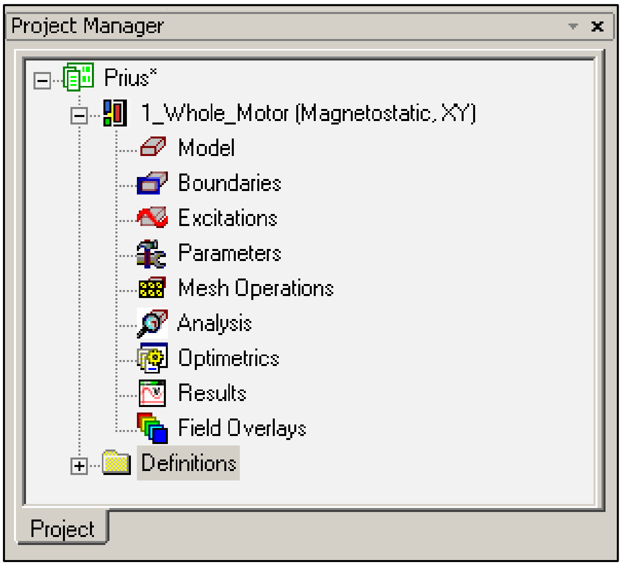
1. To access Maxwell, click the Microsoft **Start** button, select **Programs**>**Ansoft**>**Maxwell 12.**

Setting Tool Options

- **To set the tool options:**
  - **Note:** In order to follow the steps outlined in this example, verify that the following tool options are set :
  1. Select the menu item *Tools > Options > Maxwell 2D Options*
  2. Maxwell Options Window:
     1. Click the **General Options** tab
        - Use Wizards for data entry when creating new boundaries: ☑ **Checked**
        - Duplicate boundaries with geometry: ☑ **Checked**
     2. Click the **OK** button
  3. Select the menu item *Tools > Options > Modeler Options.*
  4. 3D Modeler Options Window:
     1. Click the **Operation** tab
        - Automatically cover closed polylines: ☑ **Checked**
     2. Click the **Drawing** tab
        - Edit property of new primitives: ☑ **Checked**
     3. Click the **OK** button

Opening a New Project

- **To open a new project:**
  1. In an Maxwell window, click the 🗋 icon on the Standard toolbar, or select the menu item *File > New*.
  2. Right mouse click on the project name, then select the menu item Rename. Change the project name to *Prius*
  3. Select the menu item *Project > Insert Maxwell Design*, or click on the icon
  4. Right mouse click on Maxwelldesign1 and select Rename. Change the name to *1_Whole_Motor*
  5. Click on the menu item **Maxwell 2D > Solution Type**
     - Geometry Mode: **Cartesion XY**
     - Magnetic: **Transient**



Set Model Units

- Select the menu item *Modeler > Units.* Select Units*: mm (millimeters)*

Creating the 2D Model

- Maxwell has number of **User Defined Primitives** for motor parts. These primitives can describe all the main parts of motors.

Create the Stator:

- A User Defined Primitive will be used to create the stator
- Select the menu item *Draw > User Defined Primitive > Syslib > Rmxprt > SlotCore*
- Use the values given in the panel below to create the stator

Properties: Prius_2D_v12 - 1_Whole_Motor - Modeler

Command

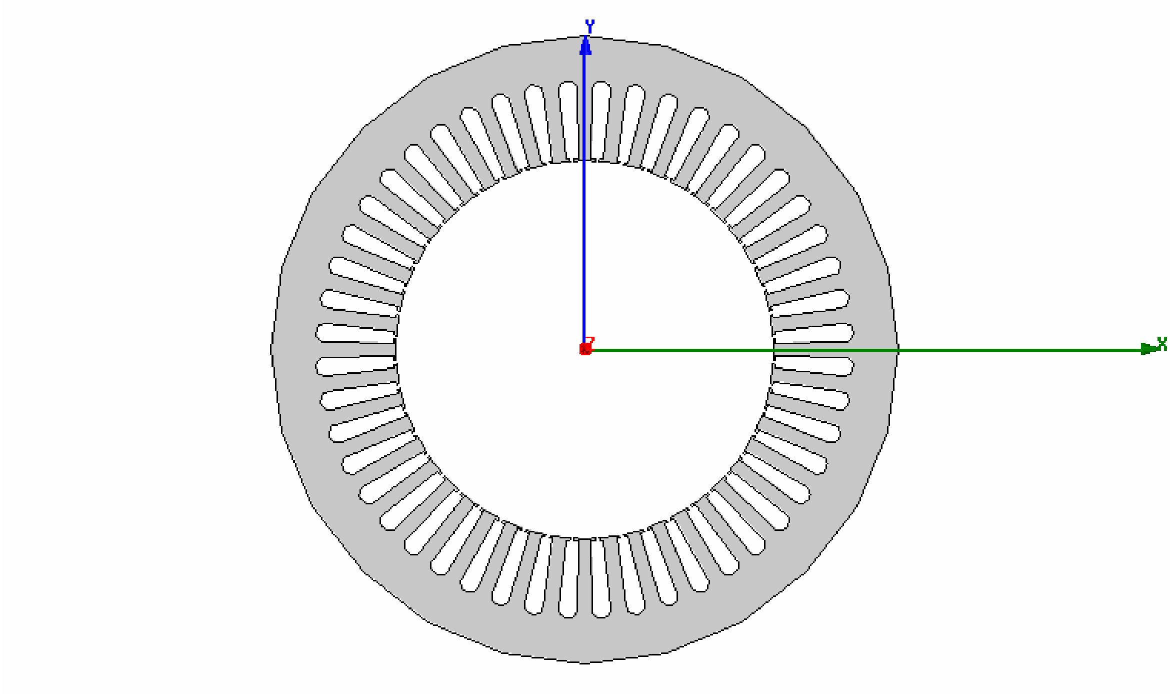
Name	Value	Unit	Evaluated Value	Description
Command	CreateUserDefinedPart			
Coordinate System	Global			
DLL Name	RMxprt/SlotCore			
DLL Location	syslib			
DLL Version	12.0			
DiaGap	161.9	mm	161.9mm	Core diameter on gap side, DiaGap<DiaYork for outer cores
DiaYoke	269.24	mm	269.24mm	Core diameter on yoke side, DiaYork<diaGap for inner cores
Length	0	mm	0mm	Core length
Skew	0	deg	0deg	Skew angle in core length range
Slots	48		48	Number of slots
SlotType	2		2	Slot type: 1 to 6
Hs0	1.03	mm	1.03mm	Slot opening height
Hs01	0	mm	0mm	Slot closed bridge height
Hs1	0	mm	0mm	Slot wedge height
Hs2	29.5	mm	29.5mm	Slot body height
Bs0	1.93	mm	1.93mm	Slot opening width
Bs1	5	mm	5mm	Slot wedge maximum width
Bs2	8	mm	8mm	Slot body bottom width, 0 for parallel teeth
Rs	5	mm	5mm	Slot body bottom fillet
FilletType	0		0	0: a quarter circle; 1: tangent connection; 2&3: arc bottom.
HalfSlot	0		0	0 for symmetric slot, 1 for half slot
LenRegion	200	mm	200mm	Region length
InfoCore	0		0	0: core; 100: region.

☐ Show Hidden

OK Cancel

Creating the 2D Model (Continued)

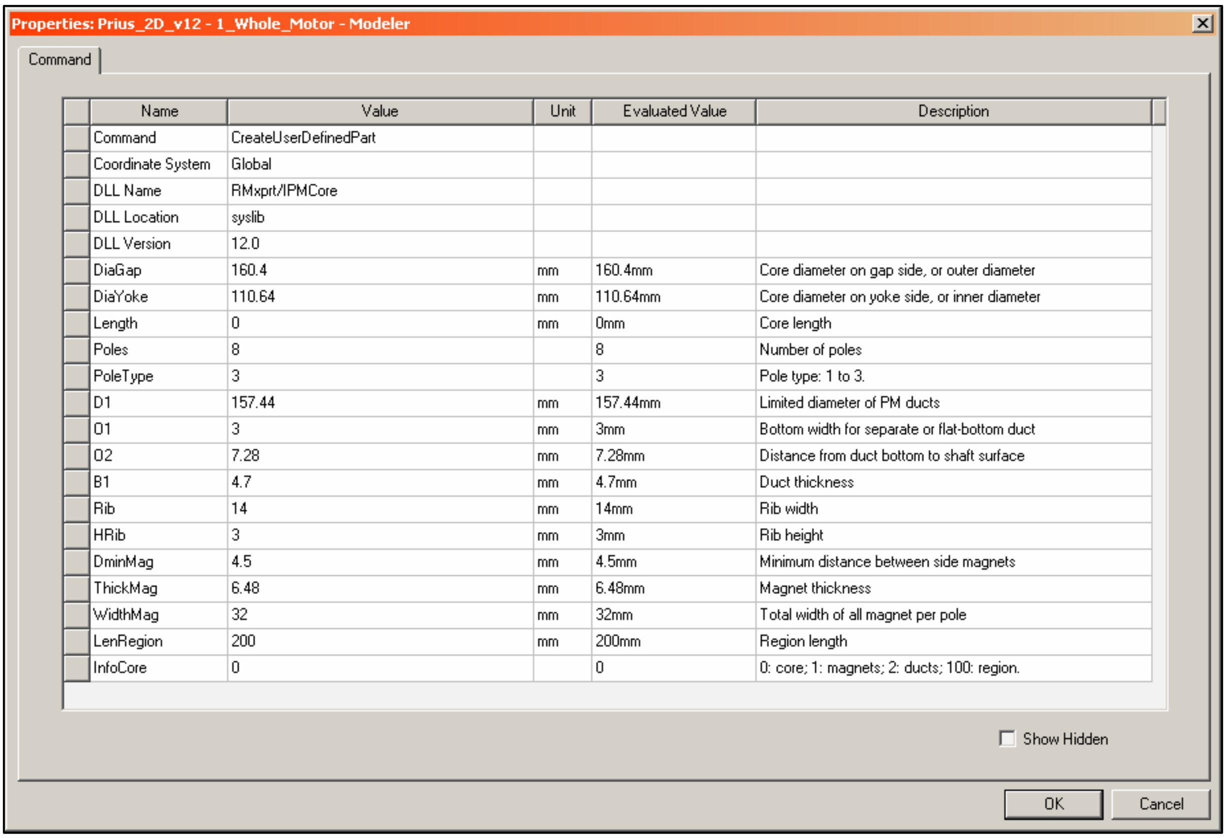
- Click on the just created object in the drawing window and in the panel on the left change its name from *SlotCore1* to *Stator*
- Note: the material will be applied afterwards



Create the Rotor

- A User Defined Primitive will be used to create the rotor
- Select the menu item *Draw > User Defined Primitive > Syslib > Rmxprt > IPMCore*
- Use the values given in the panel next page to create the rotor

Creating the 2D Model (Continued)



Properties: Prius_2D_v12 - 1_Whole_Motor - Modeler

Command

Name	Value	Unit	Evaluated Value	Description
Command	CreateUserDefinedPart			
Coordinate System	Global			
DLL Name	RMxprt/IPMCore			
DLL Location	syslib			
DLL Version	12.0			
DiaGap	160.4	mm	160.4mm	Core diameter on gap side, or outer diameter
DiaYoke	110.64	mm	110.64mm	Core diameter on yoke side, or inner diameter
Length	0	mm	0mm	Core length
Poles	8		8	Number of poles
PoleType	3		3	Pole type: 1 to 3.
D1	157.44	mm	157.44mm	Limited diameter of PM ducts
O1	3	mm	3mm	Bottom width for separate or flat-bottom duct
O2	7.28	mm	7.28mm	Distance from duct bottom to shaft surface
B1	4.7	mm	4.7mm	Duct thickness
Rib	14	mm	14mm	Rib width
HRib	3	mm	3mm	Rib height
DminMag	4.5	mm	4.5mm	Minimum distance between side magnets
ThickMag	6.48	mm	6.48mm	Magnet thickness
WidthMag	32	mm	32mm	Total width of all magnet per pole
LenRegion	200	mm	200mm	Region length
InfoCore	0		0	0: core; 1: magnets; 2: ducts; 100: region.

Show Hidden

OK Cancel

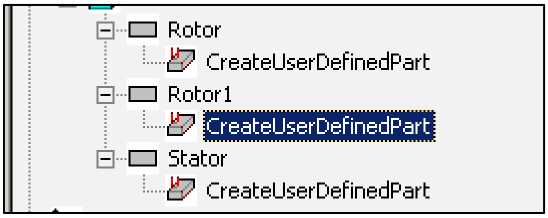
- Click on the just created object in the drawing window and in the panel on the left change its name from *IPMCore1* to *Rotor*

Create the Magnets

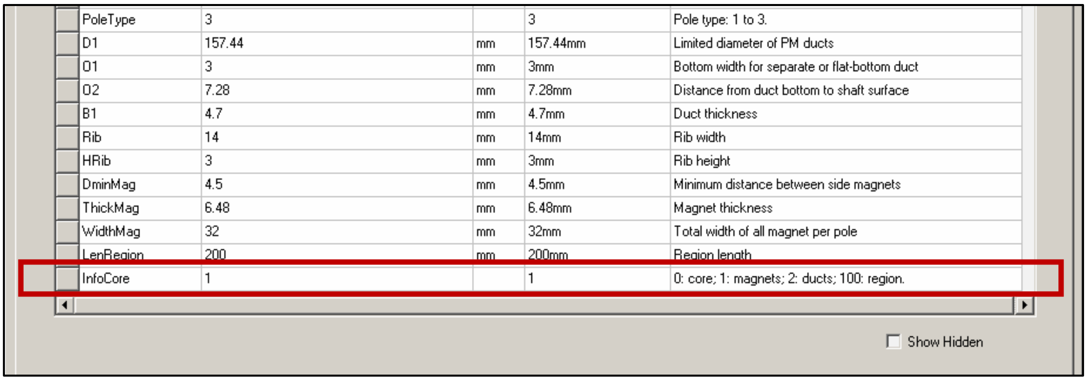
- The same User Defined Primitive can be used to create the magnets, but with different parameters. UDPs can be computed to generate different topologies.
- Select the object *Rotor*. Copy and paste the object using the *Ctrl+C*, *Ctrl+V* commands. An object *Rotor1* is created

Creating the 2D Model (Continued)

- On the modeler tree, double click on the command ‘CreateUserDefinedPart’ of the object *Rotor1*



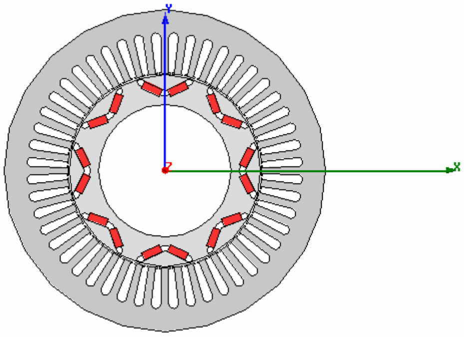
- Change the InfoCore line from 0 (Core) to 1 (Magnets)



PoleType	3		3	Pole type: 1 to 3.
D1	157.44	mm	157.44mm	Limited diameter of PM ducts
O1	3	mm	3mm	Bottom width for separate or flat-bottom duct
O2	7.28	mm	7.28mm	Distance from duct bottom to shaft surface
B1	4.7	mm	4.7mm	Duct thickness
Rib	14	mm	14mm	Rib width
HRib	3	mm	3mm	Rib height
DminMag	4.5	mm	4.5mm	Minimum distance between side magnets
ThickMag	6.48	mm	6.48mm	Magnet thickness
WidthMag	32	mm	32mm	Total width of all magnet per pole
LenRegion	200	mm	200mm	Region length
InfoCore	1		1	0: core; 1: magnets; 2: ducts; 100: region.

Show Hidden

- Change the name of the object from *Rotor1* to *Magnets*
- Change the magnets color from default to a light red.



Creating the 2D Model (Continued)

Create the Windings

- An User Defined Primitive will also be used to create the windings.
- Select the menu item *Draw > User Defined Primitive > Syslib > Rmxprt > LapCoil*
- Use the values given in the panel below to create the coil

Properties: Prius_2D_v12 - 1_Whole_Motor - Modeler

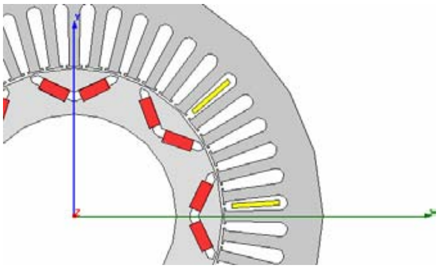
Command

Name	Value	Unit	Evaluated Value	Description
Command	CreateUserDefinedPart			
Coordinate System	Global			
DLL Name	RMxprt/LapCoil			
DLL Location	syslib			
DLL Version	12.0			
DiaGap	161.9	mm	161.9mm	Core diameter on gap side, DiaGap<DiaYork for outer cores
DiaYoke	269.24	mm	269.24mm	Core diameter on yoke side, DiaYork<diaGap for inner cores
Length	0	mm	0mm	Core length
Skew	0	deg	0deg	Skew angle in core length range
Slots	48		48	Number of slots
SlotType	2		2	Slot type: 1 to 7
Hs0	1.28	mm	1.28mm	Slot opening height
Hs1	0.1	mm	0.1mm	Slot wedge height
Hs2	29.7	mm	29.7mm	Slot body height
Bs0	2.11	mm	2.11mm	Slot opening width
Bs1	3.14	mm	3.14mm	Slot wedge maximum width
Bs2	5.65	mm	5.65mm	Slot body bottom width, 0 for parallel teeth
Rs	2.5	mm	2.5mm	Slot body bottom fillet
FilletType	0		0	0: a quarter circle; 1: tangent connection.
Layers	1		1	Number of winding layers
CoilPitch	5		5	Coil pitch measured in slots
EndExt	5	mm	5mm	One-side end extended length
SpanExt	18	mm	18mm	Axial length of end span; 0 for no span.
SegAngle	10	deg	10deg	Angle per segment of end span; <5 for true-surface end span.
LenRegion	200	mm	200mm	Region length
InfoCoil	1		1	0: winding; 1: coil; 2: terminal1; 3: terminal2; 100: region.

Show Hidden

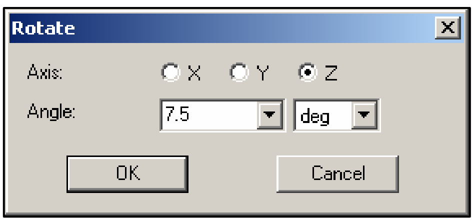
OK Cancel

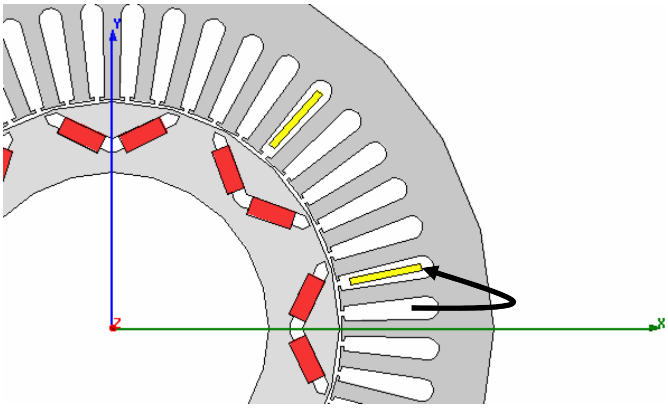
- Change the Material from **vacuum** to **Copper**
- Select the object *LapCoil1*, change its color to yellow



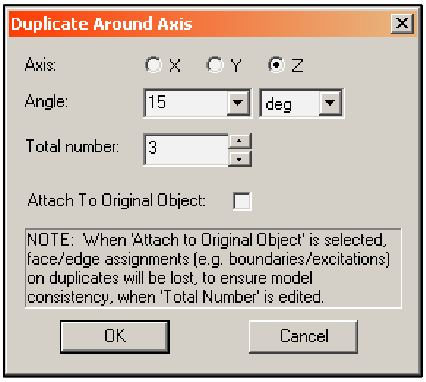
Creating the 2D Model (Continued)

- Select the object *LapCoil1*, and to apply a rotation of 7.5 deg along the Z axis, right mouse click, and select the menu item *Edit > Arrange > Rotate* or use the icon.



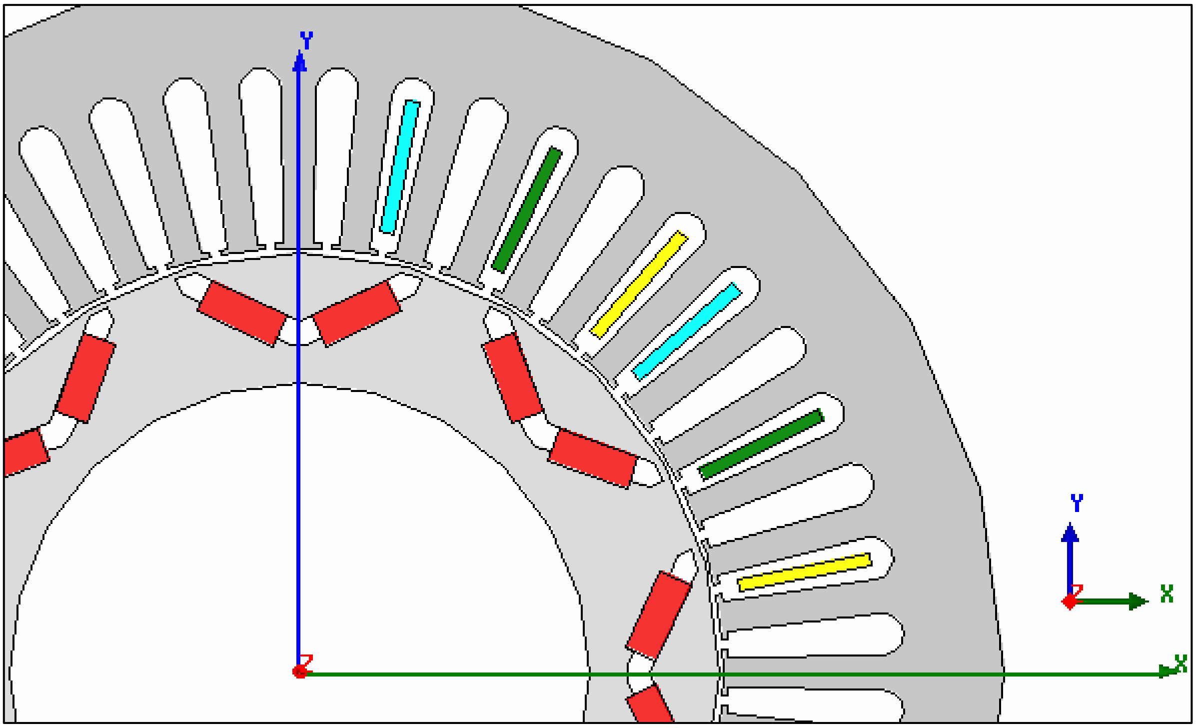


- Select the object *LapCoil1*. This coil constitutes the first coil of Phase A. We now duplicate this coil to create the first coils of Phase C and B. Right Mouse click, and select the menu item *Edit > Duplicate > Around Axis* or use the icon.

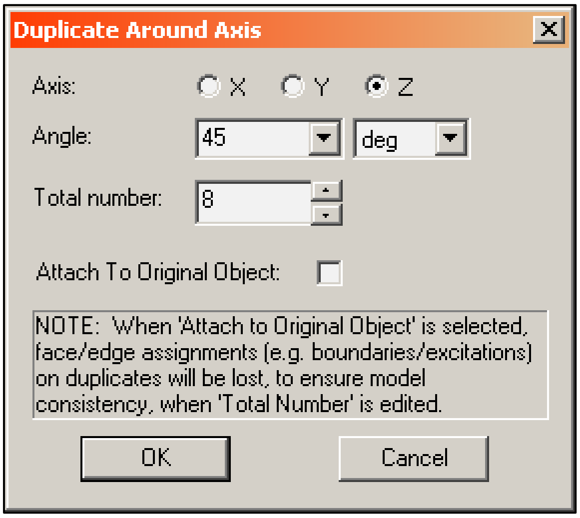


- Change the Name of objects *LapCoil1_1* and *LapCoil1_2* to *PhaseC* and *PhaseB*. Change the color of *PhaseC* to dark green and the color of *PhaseB* to light blue. Rename *Lapcoil1* to *PhaseA.*

Creating the 2D Model (Continued)

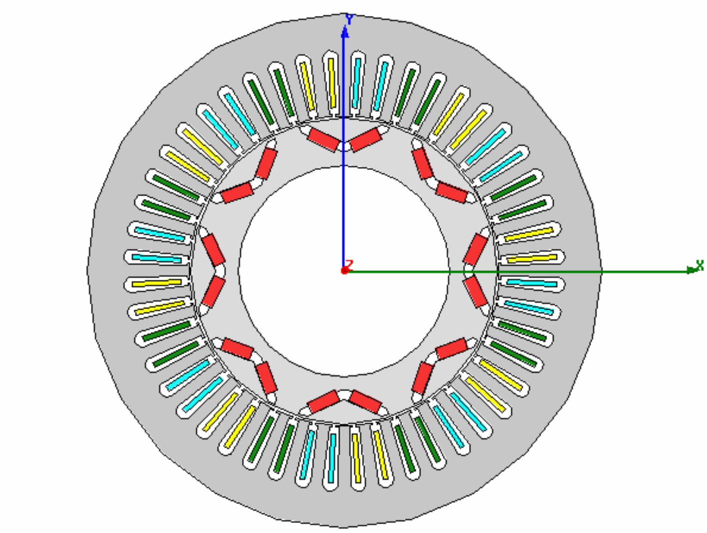


- Select the objects *PhaseA*, *PhaseB* and *PhaseC*. Right Mouse click, and select the menu item *Edit > Duplicate > Around Axis* or use the icon. Enter **45** degrees and **8** for the total number. This will create all the required coils.



Creating the 2D Model (Continued)

- The geometry of the motor is completed.

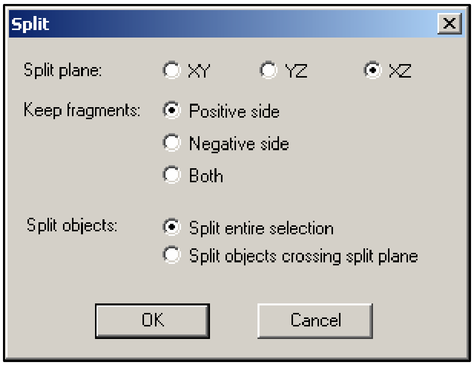


- Depending on the solver and the motor performance data that we want to look at, we might have to add more objects (for meshing or movement setting).
- Save the project. Click on the Maxwell design '*1_Whole_motor*', right mouse click and select '*Copy*'.

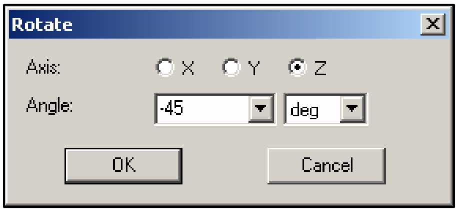
- Click on the project name, right mouse click and select '*Paste*'. Change the copied design to *2_Partial_motor*.
- We can take advantage of the topology of the motor to reduce the size of the problem. This motor has 8 pair of poles. We can only use one height of the motor. This is valid because the stator has:
  - 48 slots (8 is a divider of 48).
  - The 3-phase winding has also a periodicity of 45 degrees.
- From now on, the Maxwelldesign '*2_Partial_motor*' will be used. We have saved a copy of the whole geometry as it will be used later for other studies.

Reducing the size of the 2D Model (Continued)

- Select all the objects from the modeler tree (or you can use the *ctrl-A* command). Right mouse click and select *Edit > Boolean > Split* or use the toolbar icon . Select the *XZ* plane and keep the *positive* side.



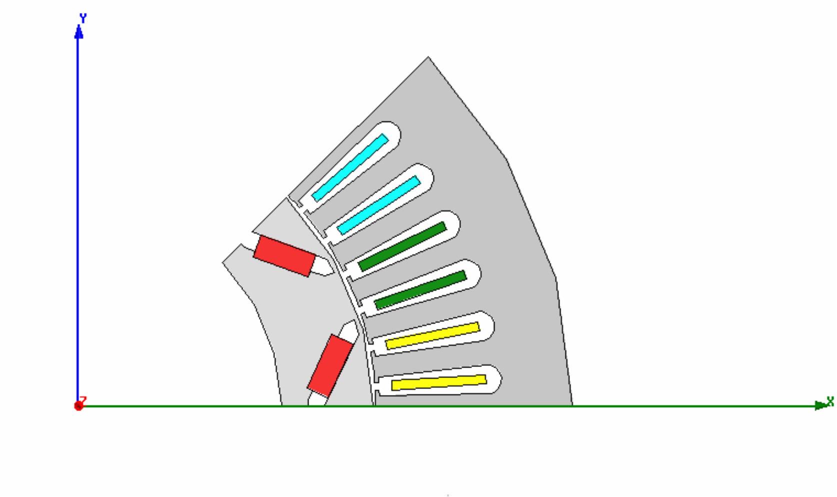
- Note: During the process, a lot of messages will appear in the dialog box. These messages inform that some objects no longer exist as they entirely lie outside the remaining model.
- We obtain half of the motor. Maintain the objects selected, right mouse click and select *Edit > Arrange > Rotate* or select the toolbar icon . Enter *-45* deg for the rotation around the *Z* axis.



- Maintain the objects selected, Right mouse click and select *Edit > Boolean > Split* or use the toolbar icon . Select the *XZ* plane and keep the *negative* side.
- Maintain the objects selected, right mouse click and select *Edit > Arrange > Rotate* or select the toolbar icon . Enter *45* deg for the rotation around the *Z* axis

Reducing the size of the 2D Model (Continued)

- The 3D model now looks like below

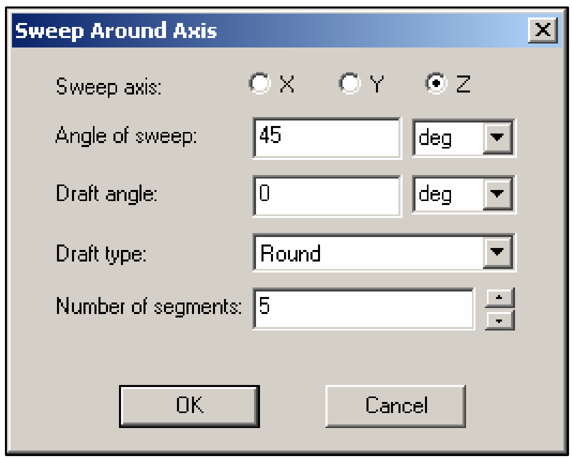


- Rename *PhaseA* to *PhaseA1* and *PhaseA_7* to *PhaseA2*. Rename *PhaseB*, *PhaseB_7*, *PhaseC* and *PhaseC_7* to *PhaseB1*, *PhaseB2*, *PhaseC1* and *PhaseC2.*
- We can now create the Region around the motor. Most of the flux is concentrated within the motor, so we do not need to have a large Region.

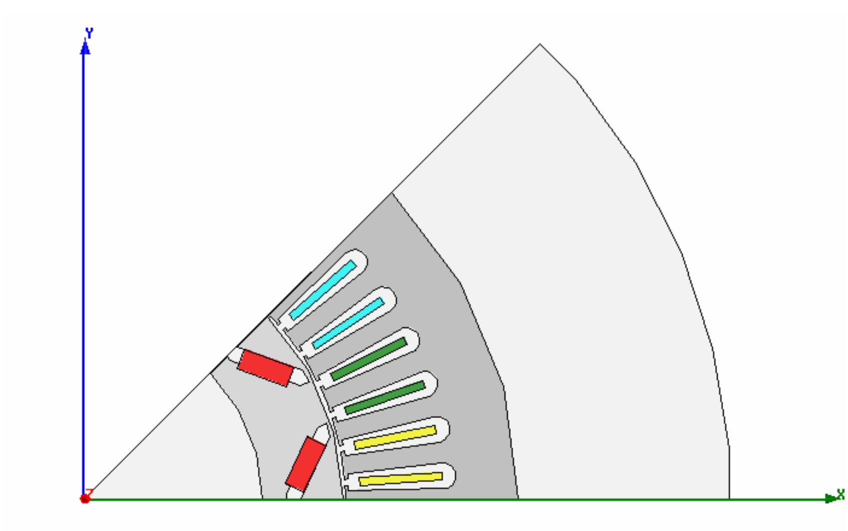
- Select *Draw > Line*
  1. Using the coordinate entry field, enter the box position
     - X: 0.0, Y: 0.0, Z: 0.0, Press the **Enter** key
  2. Using the coordinate entry field, enter the relative size of the box
     - dX: 200.0, dY: 0.0, dZ: 0.0, Press the **Enter** key
  3. Click **Enter** a second time to finish the drawing

Reducing the size of the 2D Model (Continued)

- Select *Polyline1*. Right mouse click and select *Edit > Sweep > Around Axis*.
- Enter the parameters as specified in the panel below:



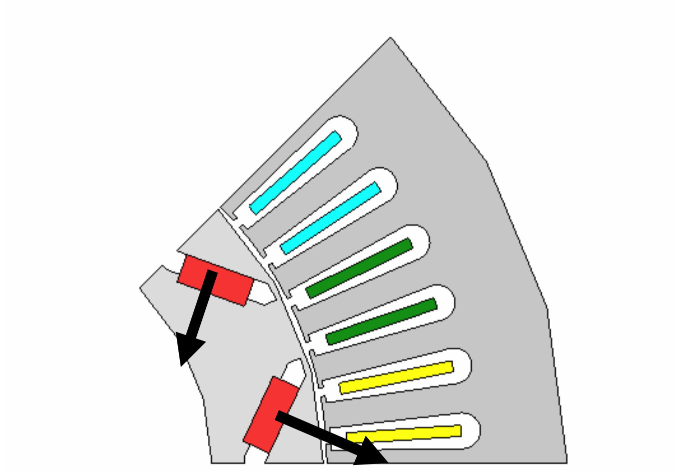
- Rename the Region from *Polyline1* to *Region*. Make sure that Vacuum is the selected material. Also, you might want to modify the render of the *Region* by increasing the transparency.



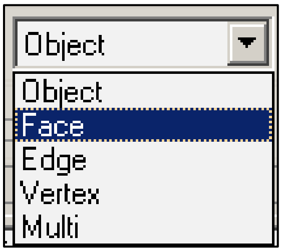
Material properties of the motor

Permanent Magnets characterization.

- The Prius Permanent Magnets (PMs) are high-strength magnets.
- In order to define PMs magnetization orientation, we need to create separate objects for each magnet. Select the object *Magnets*. Right mouse click, select *Edit > Boolean > Separate Bodies*. Rename the objects from *Magnets* to *PM1* and from *Magnets_Separate1* to *PM2*.
- Since the magnets will rotate, the orientation cannot be given through fixed coordinate systems (CS). The use of face CS is required. Face CS are CS that are attached to the face of an object. When the object moves, the Face CS also moves along with the object.
- The Prius's PMs are oriented as shown below. Therefore, we will create a face CS for each magnet.



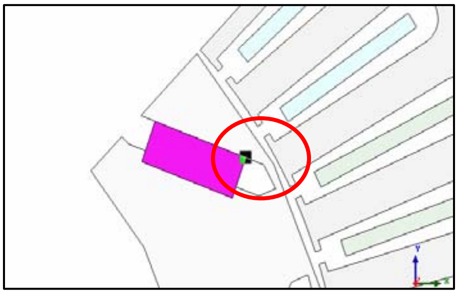
- Switch the select mode from Object to Face by clicking on the 'f' button or by using the toolbar icon:



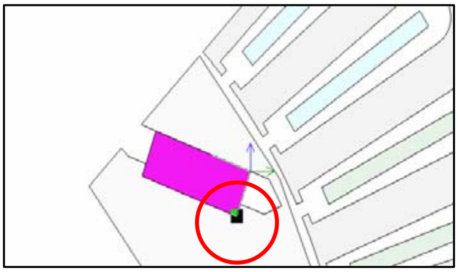
Material properties of the motor (Continued)

- Select the face of the magnet *PM1* as shown below

- To create the face CS attached to this face:
  1. Select the menu item *3D Modeler > Coordinate System > Create > Face CS* or select the toolbar icon
  2. The modeler is in draw mode. It expects the center of the face CS that has to be on the selected plane to be selected. Snap the mouse pointer to one of the corner of the face, using the "*snap to vertex symbol*" . This defines the CS center.

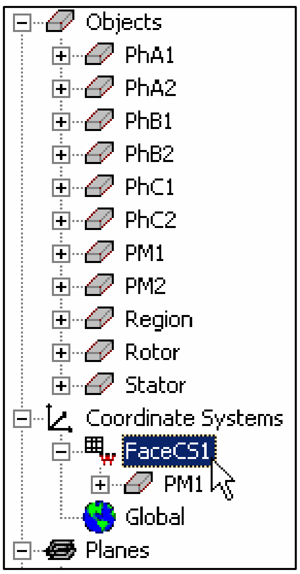


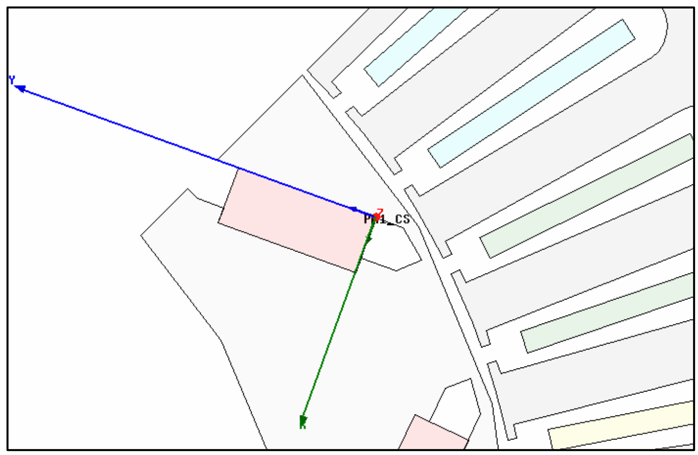
  3. You need to enter the direction of the X axis. Snap the mouse point at another vertex of the face as shown below



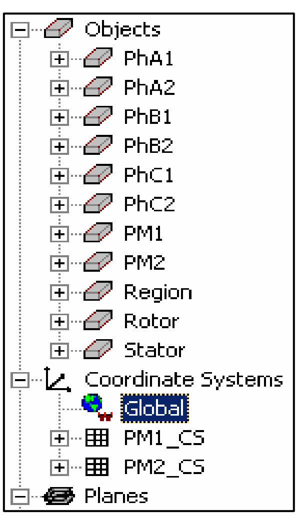
Material properties of the motor (Continued)

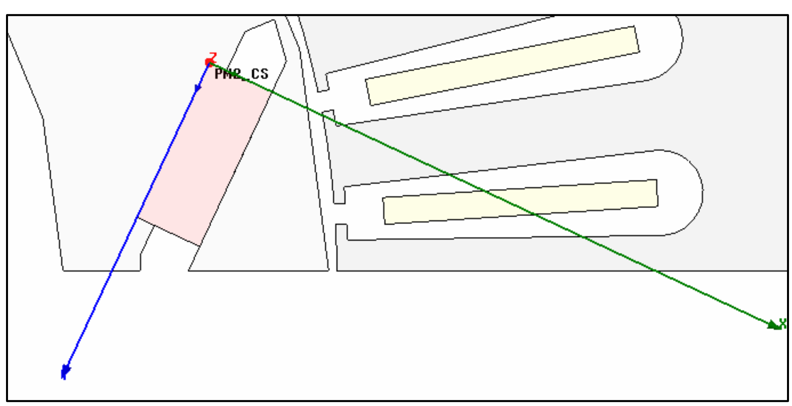
- The face CS is created. Its default name is *FaceCS1*. Change its name from *FaceCS1* to *PM1_CS*.



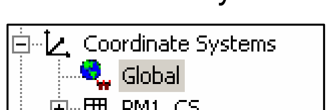


- Repeat the same operation to create the face CS *PM2_CS* attached to *PM2*. Make sure to have the X axis looking toward the air gap



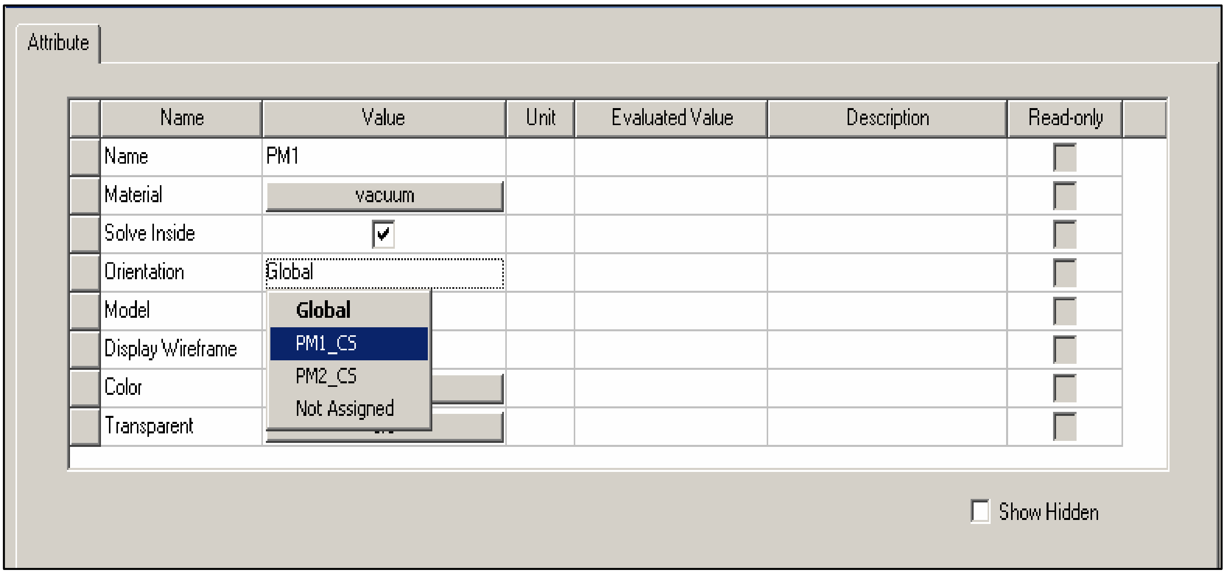


- Reset the working CS to the Global CS by clicking on *Global* as shown below.

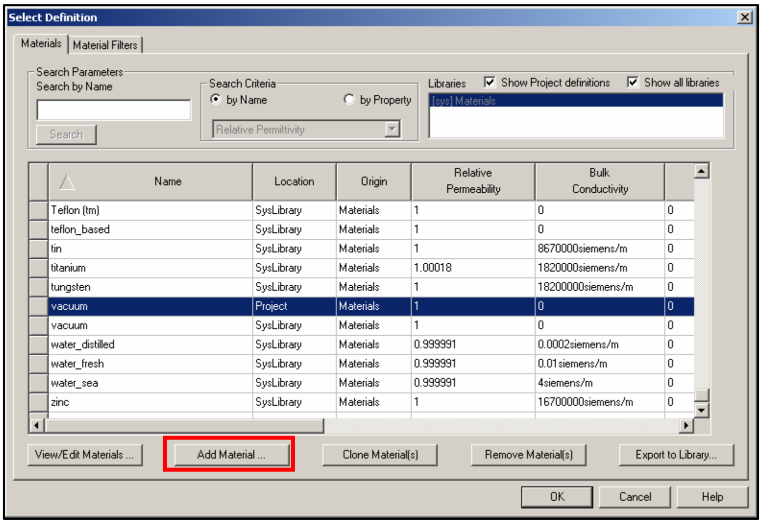


Material properties of the motor (Continued)

- Edit the attributes of the object *PM1*. Modify the Orientation of the object by selecting the *PM1_CS* coordinate system. This CS will be the reference for the magnetization direction.

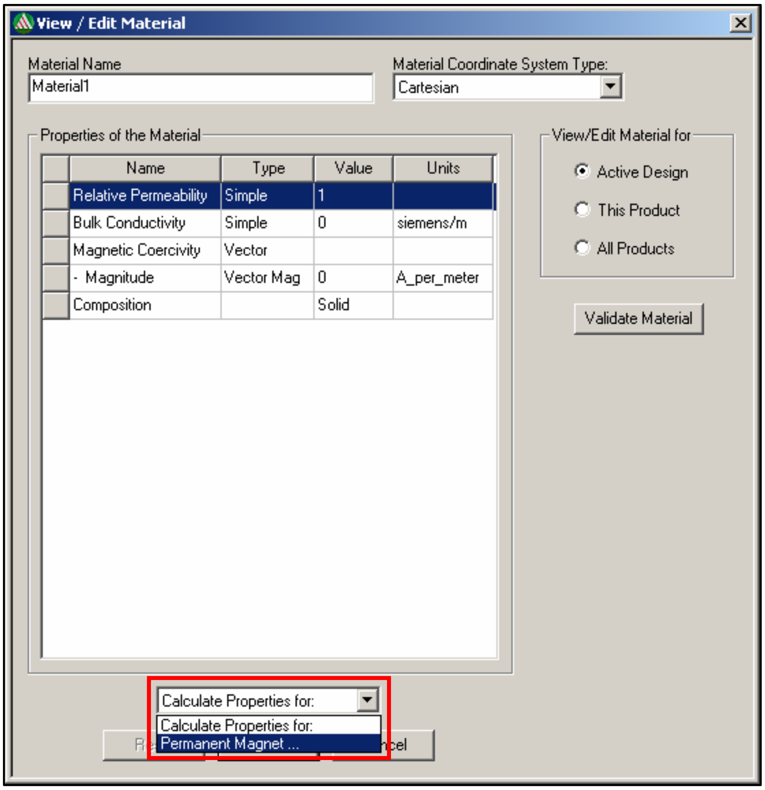


- To enter into the material database, click on the Material button (the default material is *Vacuum*). The Prius magnet is not part of the default library, so click on the Add material button

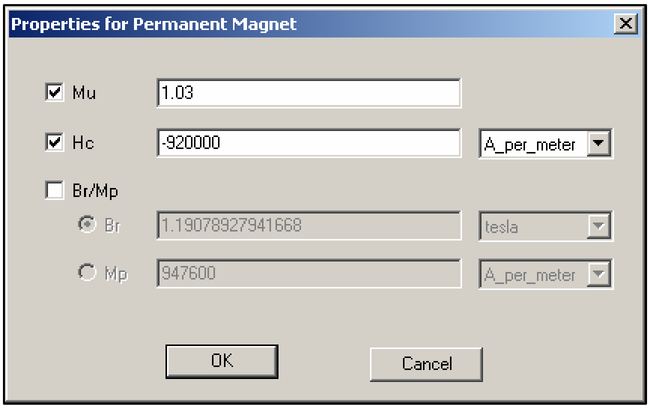


Material properties of the motor (Continued)

- We have a special menu to enter Permanent Magnet parameters. At the bottom of the View/Edit material window, select the "*Permanent Magnet*" entry.

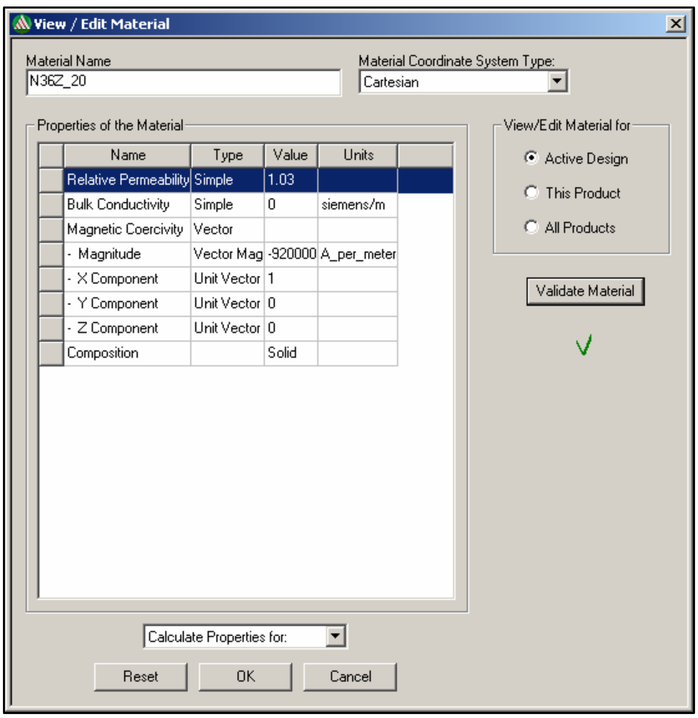


- Enter the values given below to define the magnet strength



Material properties of the motor (Continued)

- Change the material name to *N36Z_20*.
- If the coordinate system *PM1_CS* is such that the **X axis** goes in the opposite direction of the air gap accordingly to the image below, leave the X orientation to 1 and 0 for the Y and Z components. If the X axis was in the opposite direction, you would need to enter -1 for the X component.

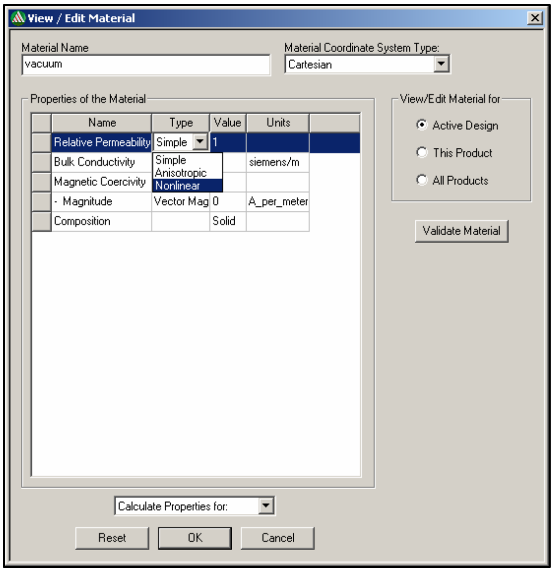


- Click on the Validate button before closing the window to check the material definition.
- Edit the attributes of the object *PM2*. Modify the Orientation of the object by selecting the *PM2_CS* coordinate system. This CS will be the reference for the magnetization direction. If the definition of PM2_CS is consistent with PM1_CS (X axis in the direction of the air gap), you can use the same material for *N36Z_20* for *PM2*. If it is not the case, you can clone the material *N36Z_20* and change the orientation to be consistent with the *PM2_CS* axis.

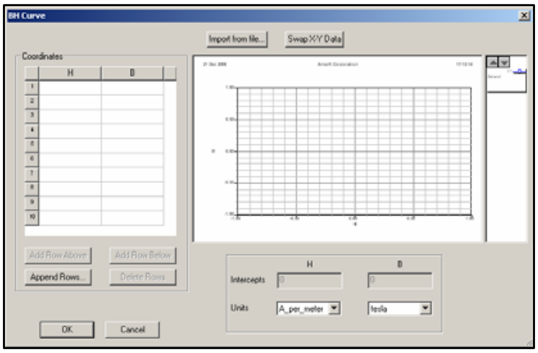
Material properties of the motor (Continued)

Steel definition

- The stator and rotor shares the same material. Select the objects *Stator* and *Rotor*. Edit their attributes, change the affected material. In the material database, add a new material called *M19_29G.*
- The steel is non linear. To enter the non-linear B-H Characteristic, change the Relative Permeability from “Simple” to “Nonlinear”



- Click on the BH curve button in the Value column. The BH curve entry window appears



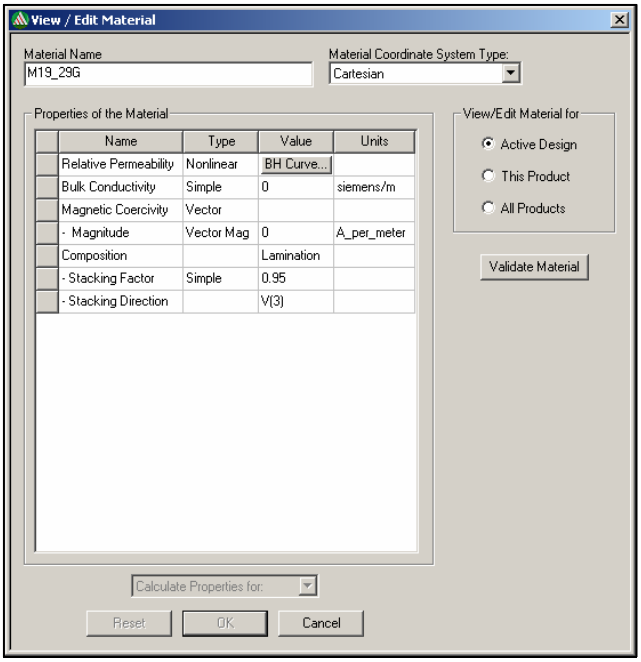
Material properties of the motor (Continued)

- Enter the B-H characteristics with the values given below

H	B
0	0
22.28	0.05
25.46	0.1
31.83	0.15
47.74	0.36
63.66	0.54
79.57	0.65
159.15	0.99
318.3	1.2
477.46	1.28
636.61	1.33
795.77	1.36
1591.5	1.44
3183	1.52
4774.6	1.58
6366.1	1.63
7957.7	1.67
15915	1.8
31830	1.9
111407	2
190984	2.1
350138	2.3
509252	2.5
560177.2	2.563994494
1527756	3.779889874

Material properties of the motor (Continued)

- We neglect the Eddy current in this example, therefore we leave the conductivity to 0.
- Validate the material before exiting the View/Edit material window

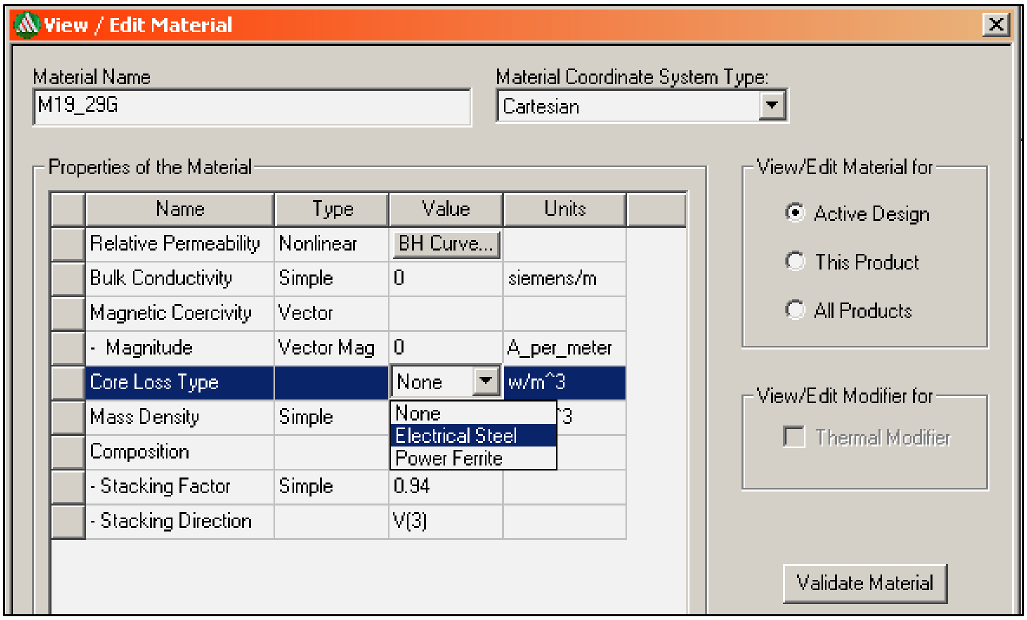


- Make sure that *M19_29G* is affected to the Rotor and Stator.

Coreloss

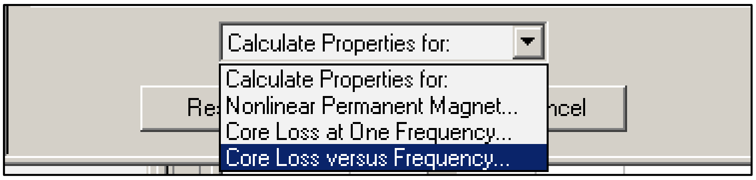
- This section is only necessary if you wish to compute the coreloss of the motor.
- In the Transient solver, we are able to compute coreloss (or hystereris loss), stranded loss and eddy current loss (or proximity loss). We will only consider coreloss in this document.
- We need to enter the loss values of the steel. A dedicated menu enables the user to enter the data.
- Extend the project tree, and double click the Material definition of the Steel M19_29G

- Use the pull down menu to enable core loss for *Electrical Steel* material

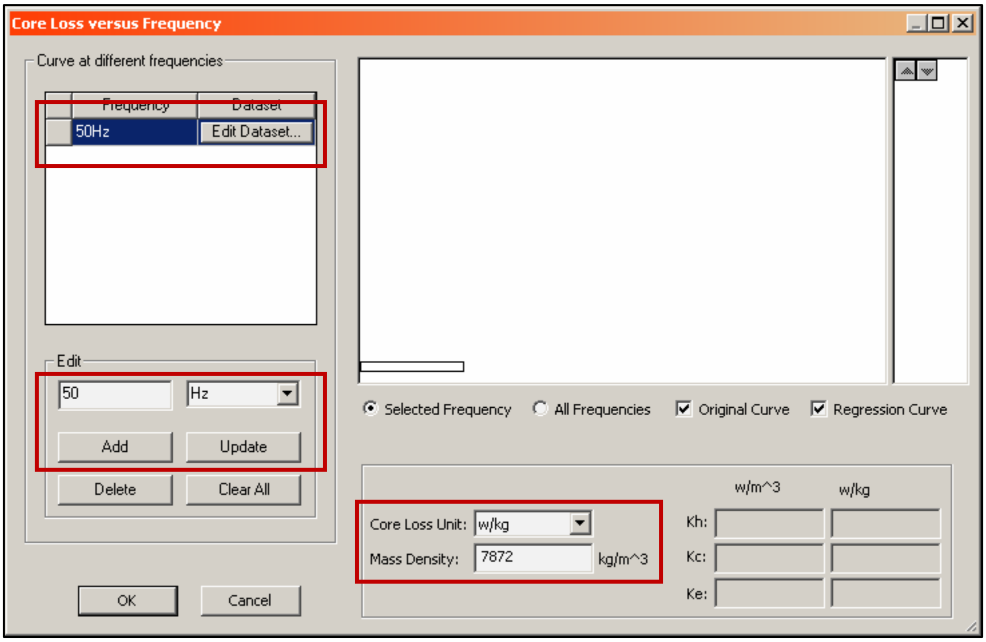


Coreloss (Continued)

- The Maxwell solver requires the coefficients Kh, Kc, Ke and Kdc. A special menu allows the coefficients to be derived from manufacturer core loss data
- Select at the bottom of the material definition window from the pull down menu Core Loss versus Frequency



- The Core Loss versus Frequency menu pops up. We provide the data for several frequencies:
  1. Select *W/kg* for the Core Loss Unit
  2. Enter *7872 kg/m^3* for the Mass density of the Steel
  3. Enter **50 Hz** in the Edit window
  4. Click on *Add*
  5. Click on *Edit Dataset* in the Frequency Window

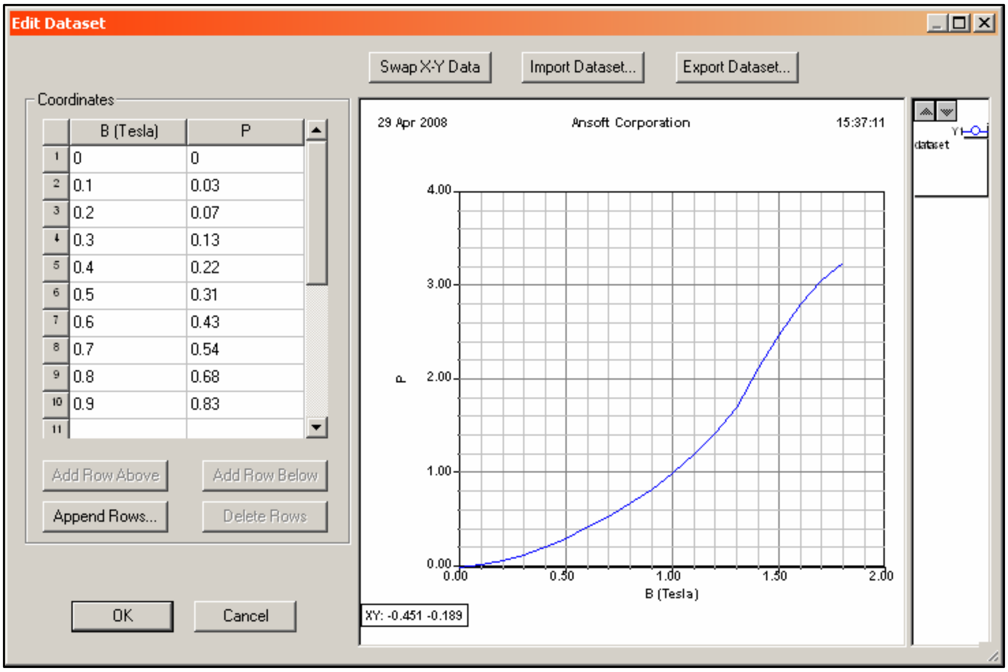


Coreloss (Continued)

- Enter the loss curve at 50 Hz:

50 Hz

B	P
0	0
0.1	0.03
0.2	0.07
0.3	0.13
0.4	0.22
0.5	0.31
0.6	0.43
0.7	0.54
0.8	0.68
0.9	0.83
1	1.01
1.1	1.2
1.2	1.42
1.3	1.7
1.4	2.12
1.5	2.47
1.6	2.8
1.7	3.05
1.8	3.25



- Accept the setting

- Using the same method enter the loss curves for 100, 200, 400, 1000 Hz

100Hz

B	P
0	0
0.1	0.04
0.2	0.16
0.3	0.34
0.4	0.55
0.5	0.8
0.6	1.08
0.7	1.38
0.8	1.73
0.9	2.1
1	2.51
1.1	2.98
1.2	3.51
1.3	4.15
1.4	4.97
1.5	5.92

200Hz

B	P
0	0
0.1	0.09
0.2	0.37
0.3	0.79
0.4	1.31
0.5	1.91
0.6	2.61
0.7	3.39
0.8	4.26
0.9	5.23
1	6.3
1.1	7.51
1.2	8.88
1.3	10.5
1.4	12.5
1.5	14.9

400Hz

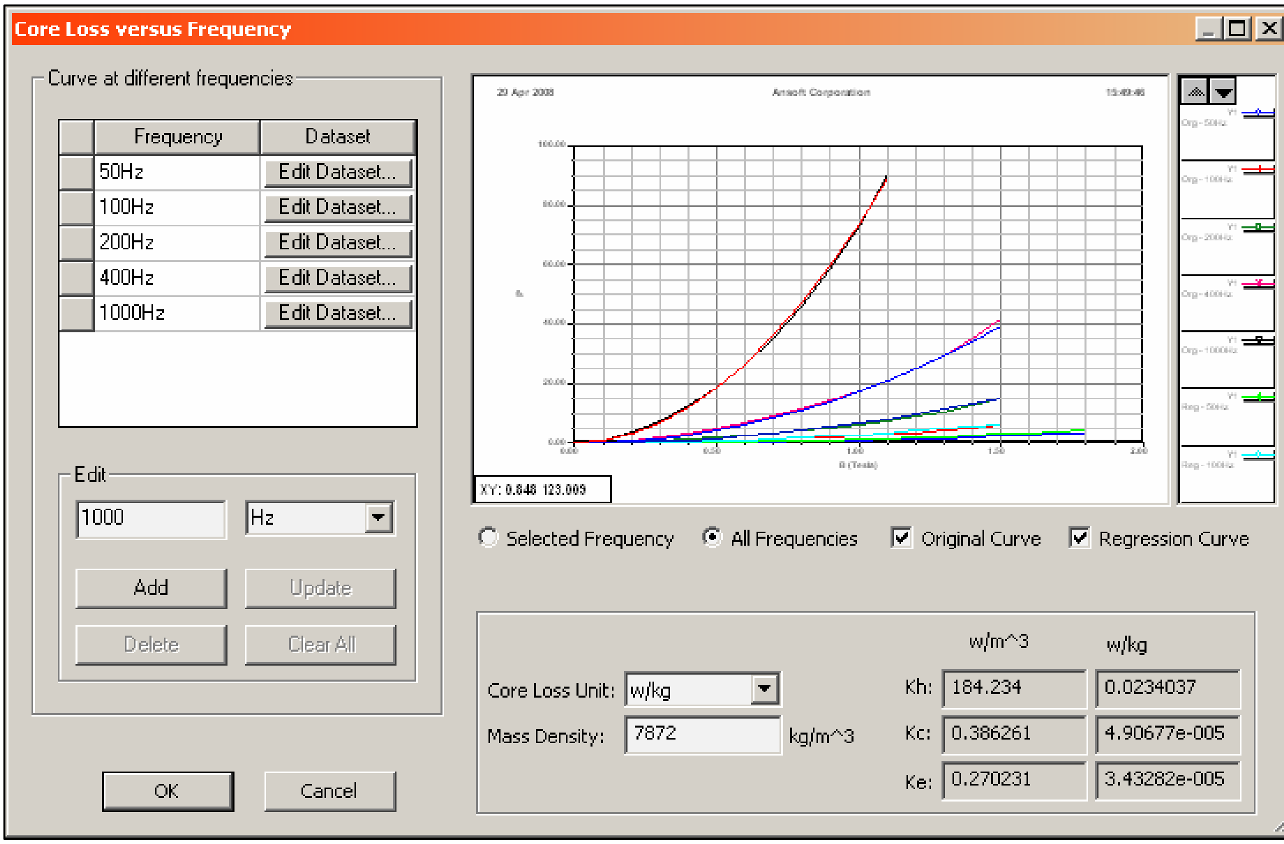
B	P
0	0
0.1	0.21
0.2	0.92
0.3	1.99
0.4	3.33
0.5	4.94
0.6	6.84
0.7	9
0.8	11.4
0.9	14.2
1	17.3
1.1	20.9
1.2	24.9
1.3	29.5
1.4	35.4
1.5	41.8

1000Hz

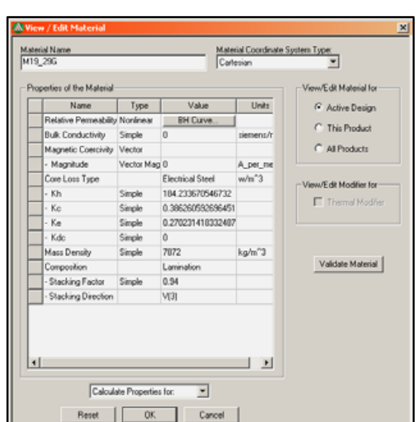
B	P
0	0
0.1	0.99
0.2	3.67
0.3	7.63
0.4	12.7
0.5	18.9
0.6	26.4
0.7	35.4
0.8	46
0.9	58.4
1	73
1.1	90.1

Coreloss (Continued)

- The coreloss coefficient are automatically calculated

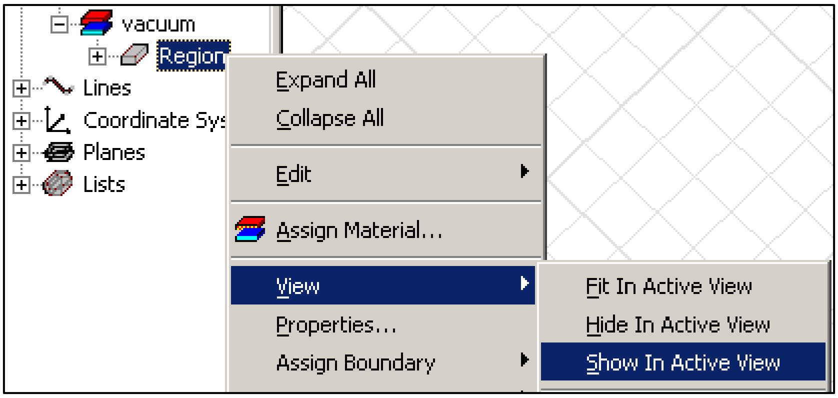


- Accept the setting. The material definition now includes the coreloss coefficients

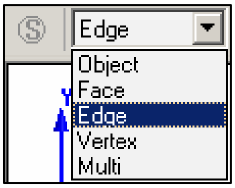


Applying Master/Slave Boundary Condition

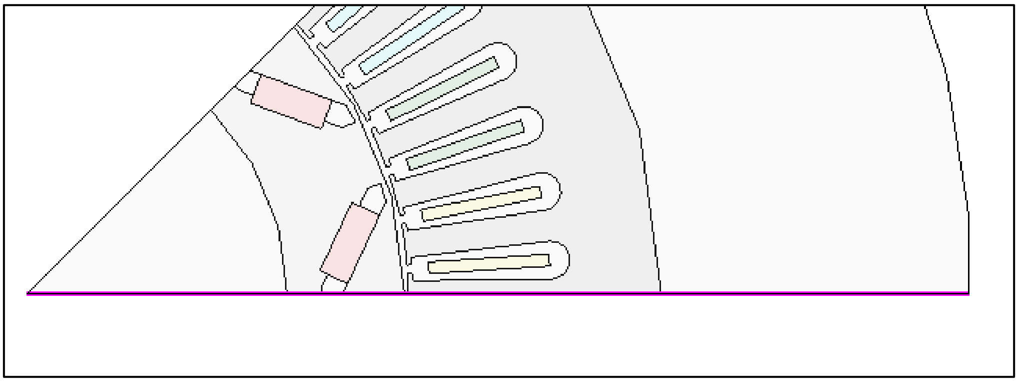
- The Master and Slave boundary condition takes advantage of the periodicity of the motor. Two planes are to be defined: the master and slave planes. The H-field at every point on the slave surface matches the (plus or minus) H-field at every point on the master surface.
- Select the object *Region* from the active view. Right mouse click, then select *View> Show In Active View* as shown below



- Change the Select mode to *Edge*

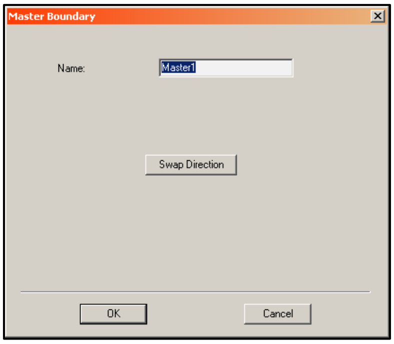


- Select one of the bounding line of the *Region*

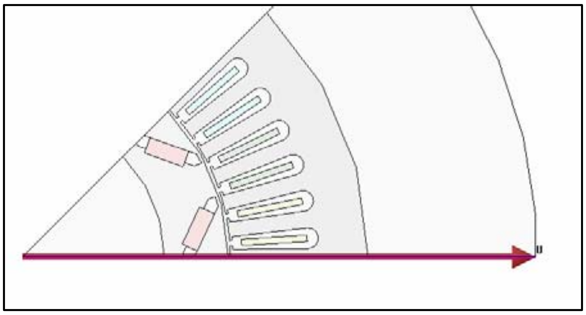


Applying Master/Slave Boundary Condition (Con'd)

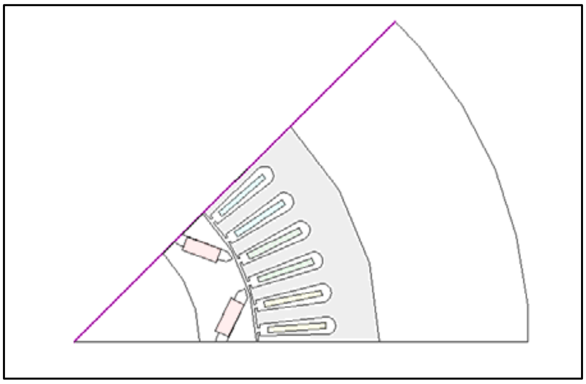
- Right mouse click, select *Assign Boundary > Master*



- The vector u is defined correctly. Accept the setting.

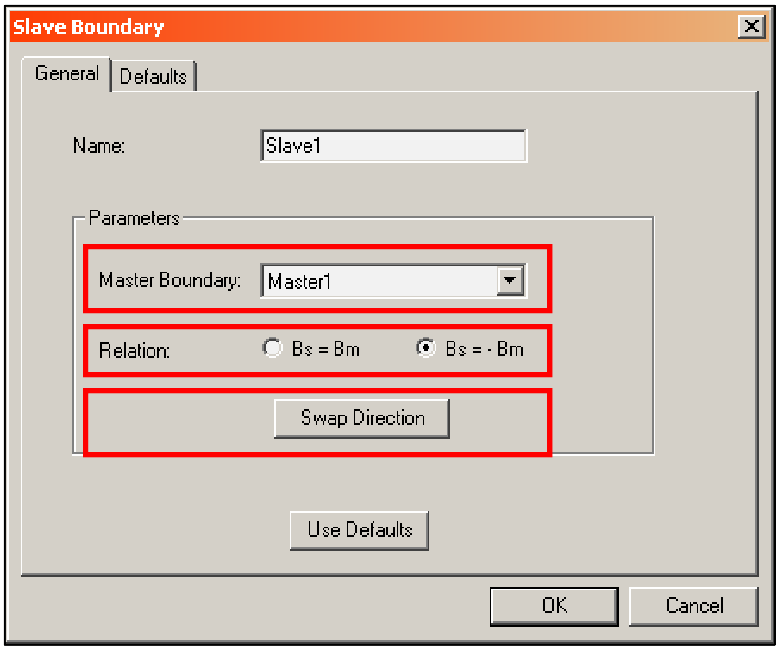


- Select the opposite edge of the Region

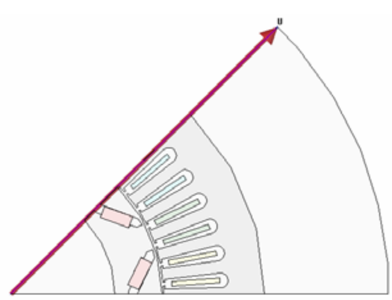


Applying Master/Slave Boundary Condition (Cont'd)

- Right mouse click, select *Assign Boundary > Slave*

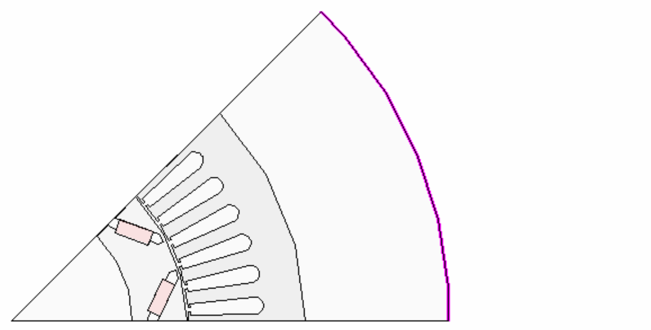


1. We first need to give the reference of the master condition. For the Master Boundary, since we haven't changed the default name, Select *Master1*
2. Select *Swap direction* for the u vector definition if the vector u does not have the same direction than the u vector of the Master condition.
3. The model represents one pole out of height. Since we represent an odd number of poles, the condition at the slave surface is Slave = -Master
4. Accept the set up



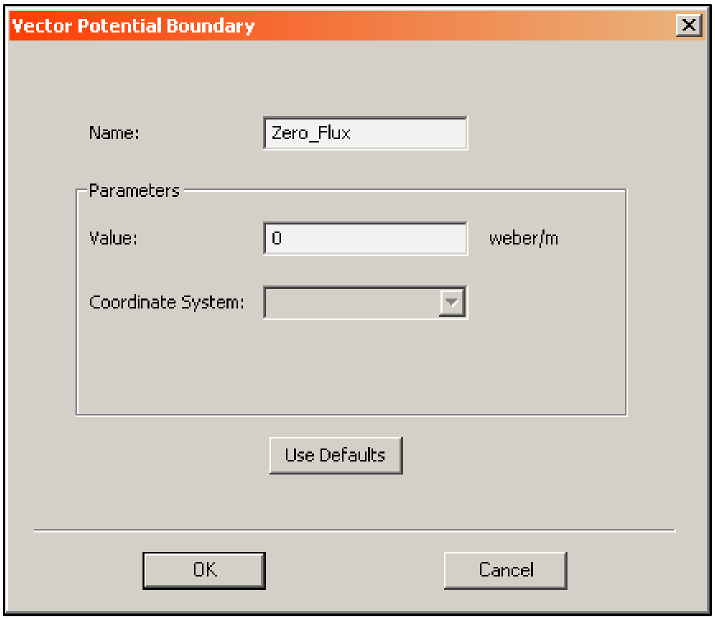
Applying Zero Vector Potential Boundary Condition

- At the limit of the *Region*, select the five segments of the outside limit of the *Region*. Use the Ctrl button to allow multi selections



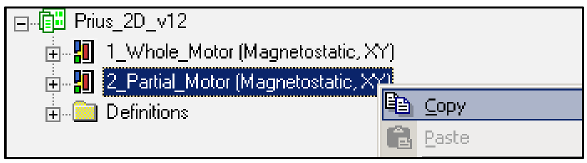
- Right Mouse Click, Select *Assign Boundary > Vector Potential*

1. Put *0* Weber/m for the value
2. Name the condition *Zero_Flux*



STATIC ANALYSIS

- We will study the different static parameters of the motor.
- Save the project. Click on the Maxwell design '*2_Partial_motor*', right mouse click and select 'Copy'.

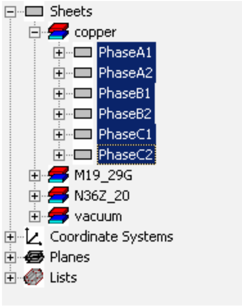


- Click on the project name, right mouse click and select Paste. Change the copied design to '*3_Partial_motor_MS*'.

No Load Study

- The first analysis that will be performed consists in computing the fields due to the permanent magnets.

- The Coils are not needed in the model since no current is defined. Select the 6 coils. Then, Uncheck the radio button "Model" from the property window. Note that the Name of object line is empty since we have selected several objects.

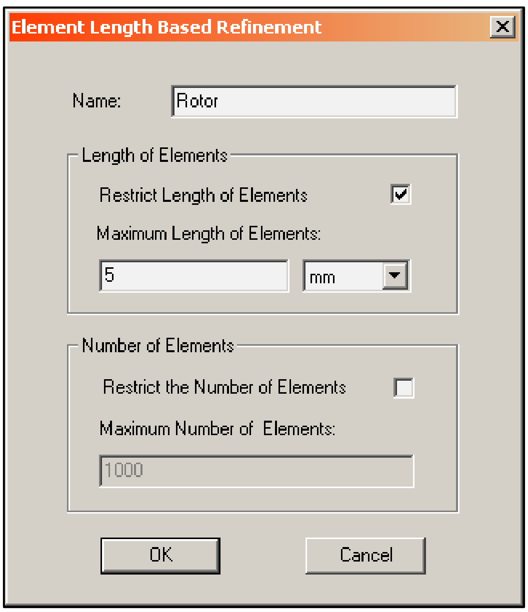


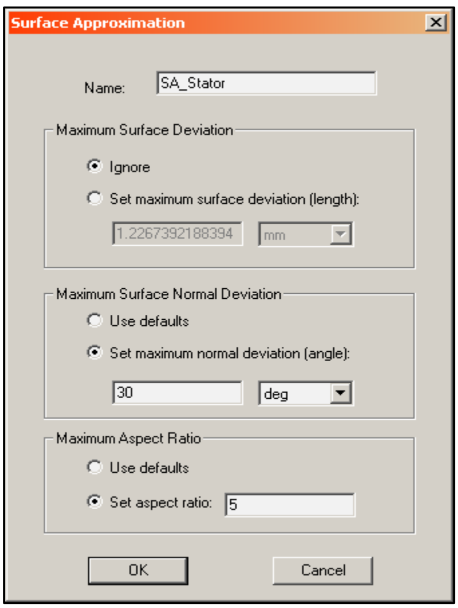
Name	Value	Unit	Evaluated Value
Name			
Orientation	Global		
Model	☐		
Display Wi...	☐		
Color	Edit		
Transparent	0		

- Leave the Coils selected, and Hide the coils by selecting the menu item **View > Hide Selection > Active view** or using the toolbar button

Apply Mesh Operations

- The adaptive meshing is very effective, so it is not necessary to enter dedicated mesh operations. However, it is always a good idea to start with a decent initial mesh in order to reduce time computation since we know where the mesh needs to be refined for a motor. The non linear resolution will be faster with a small aspect ratios for the elements in the steel.
- Select the **Rotor**. Right Mouse Click and Select *Assign Mesh Operation > Inside Selection > Length Based*



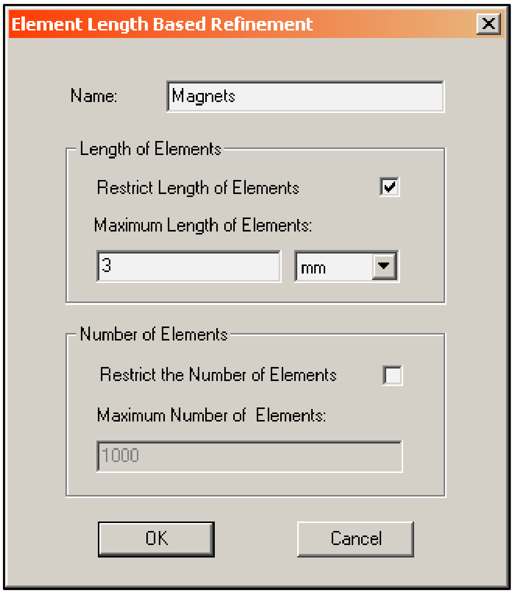


- Restrict the length of elements to **5 mm.**
- Rename the mesh operation *Rotor*

- Select the **Stator**. We want to minimize the number of elements for the curved line of the slots. Right Mouse Click and Select **Assign Mesh Operation > Surface Approximation**.
  - Input **30deg** for the Maximum surface deviation
  - Select **5** for the Maximum aspect Ratio.
  - Rename the mesh operation *SA_Stator*

Apply Mesh Operations (Continued)

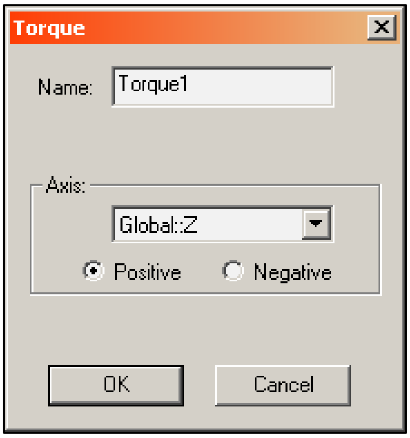
- Select **PM1** and **PM2**. Right Mouse Click and Select *Assign Mesh Operation > Inside Selection > Length Based*



  - Restrict the length of elements to **3** mm.
  - Rename the mesh operation **Magnets**

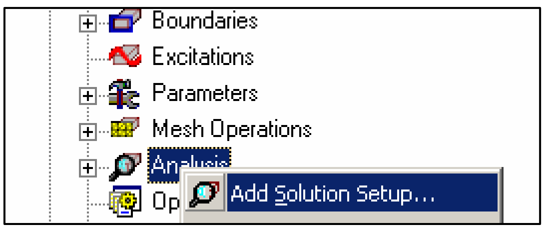
Apply Torque computation

- Select the objects *PM1*, *PM2* and *Rotor*. Right mouse click and select *Assign Parameters > Torque*

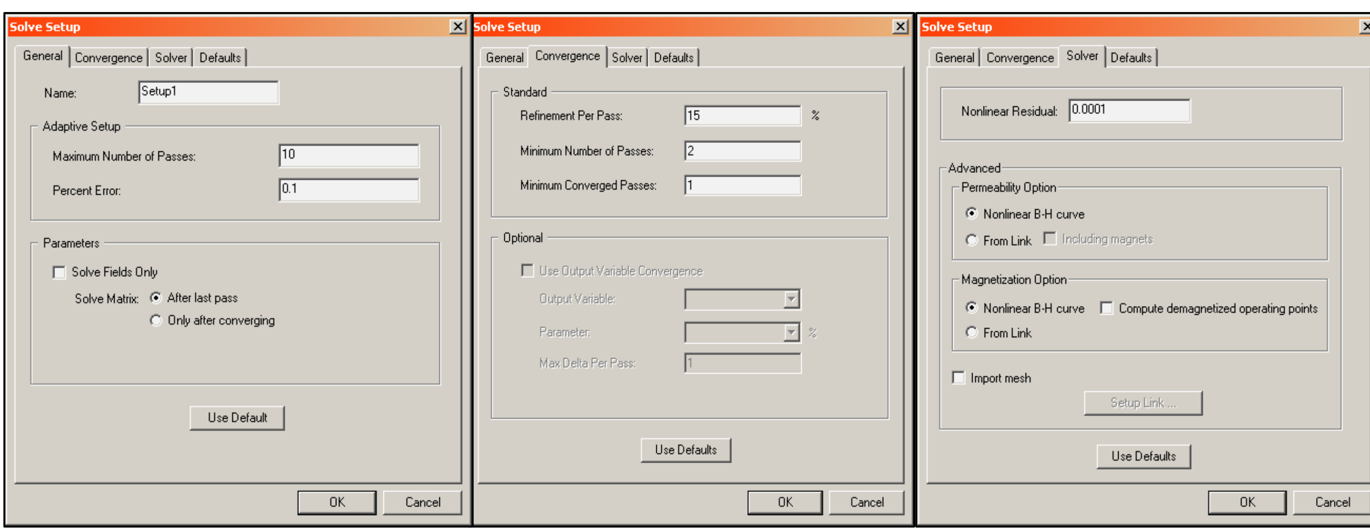


Add an Analysis Setup

- From the project manager, right mouse click on *Analysis* and select *Add Solution Setup*:



1. Enter 10 for the maximum number of passes
2. Enter 0.1% for the error
3. In the convergence panel, enter 15% for the refinement
4. Make sure that the Non Residual is set to 0.0001%. Click Ok to record the analysis setup

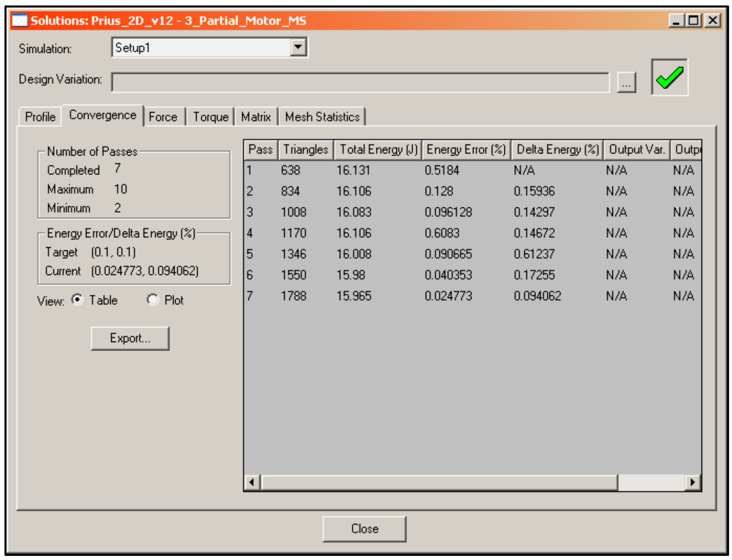


Analyse

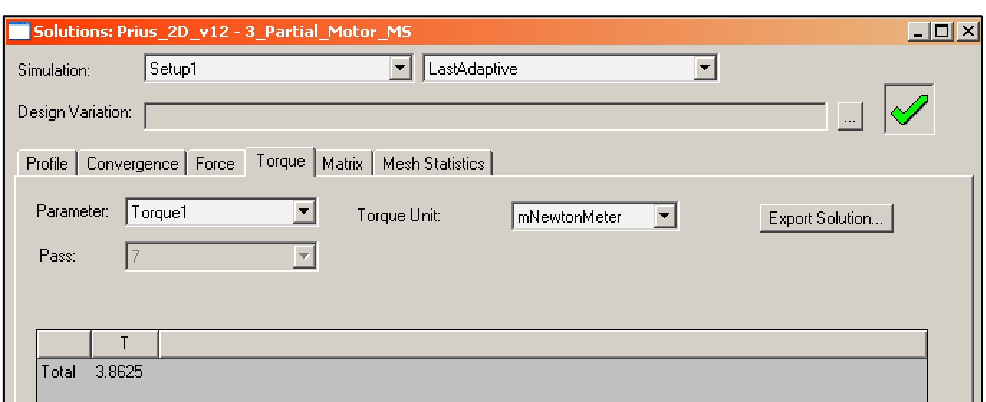
- Right mouse click on the setup et select Analyze or click on the  icon.

Post processing

- The computation takes 7 passes to converge. The Convergence panel can be seen by right mouse clicking on Setup1, selecting the menu item *Convergence*

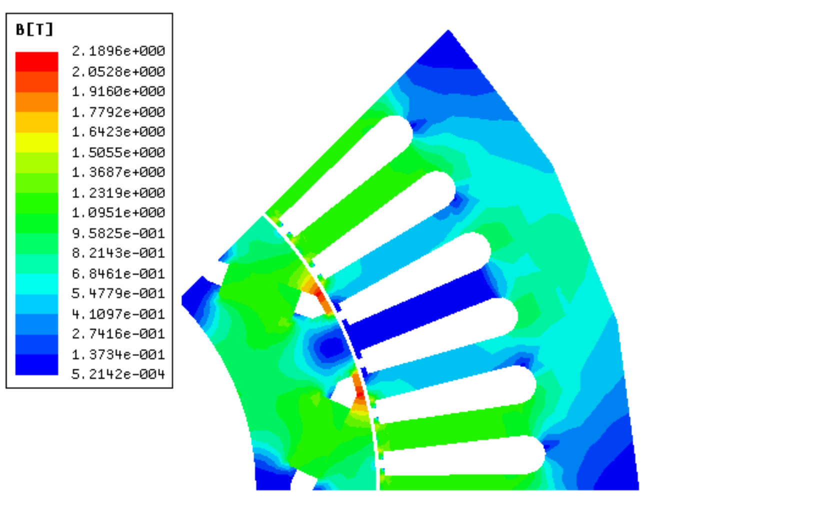


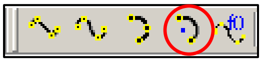
- **Torque value.** Select the Solutions tab, the torque is given for a one meter depth motor. The torque for the full motor needs to be multiplied by 8 (symmetry factor), then by 0.082 (to account for the motor length). This gives 2.5mN.m, which sounds reasonable: the value is very small in regards to the full load operation. Different angles between the rotor and the stator would give different values.

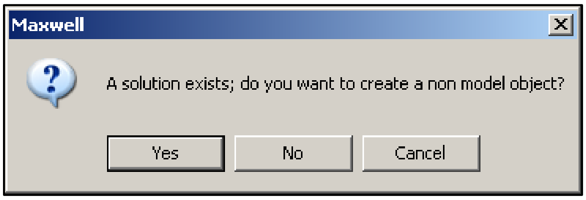


Post Processing (Cont'd)

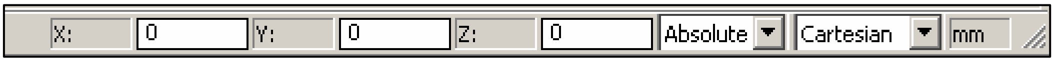
- **Plot magnetic flux density**. Select the *Rotor*, *Stator*, *PM1, PM2* right mouse click, select *All Object Faces*. Right mouse click again and select *Fields > B > Mag_B.* We obtain the distribution of **B** on the objects. The steel is highly saturated close to the magnets as expected. This saturations appears just because of the magnets strengths.



- **Plot the magnetic flux strength H in the air gap**. We need to draw a post-processing line to view the field:
  1. Draw an arc. Select the menu item Draw > Arc > Center Point or use the corresponding toolbar icon 
  2. Accept to continue to draw a non model object. This will not invalidate the existing solution

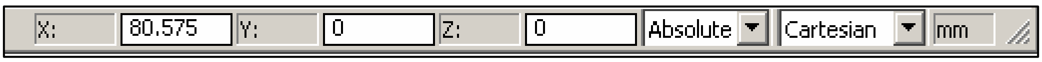


  3. Enter the center of the arc: *0,0,0 mm* and hit enter

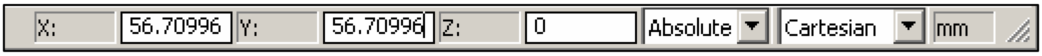


Post Processing (Cont'd)

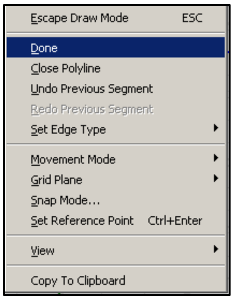
4. Enter the first point of the arc. This point is at the middle of the air gap on the YZ plane. Enter *80.575, 0 , 0 mm* and hit enter.



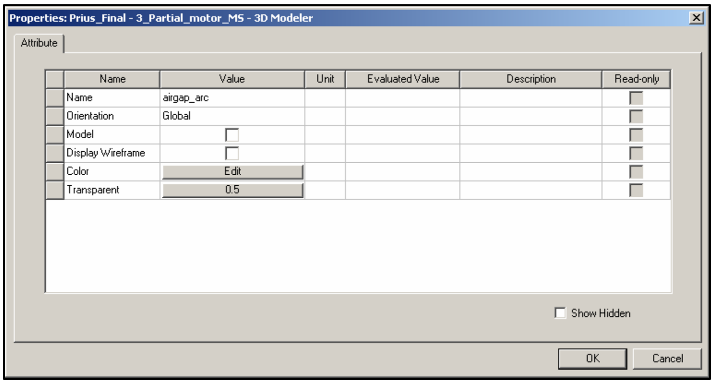
5. Enter the last point of the arc. This point lies on the plane XY, with a 45° angle with the X- axis. 80.575/ √2= 56.70996(…). Enter *56.70996, 56.70996, 0 mm* and hit enter*.*



6. To finish the arc, move the mouse on the drawing area, right mouse click, and select the menu entry done

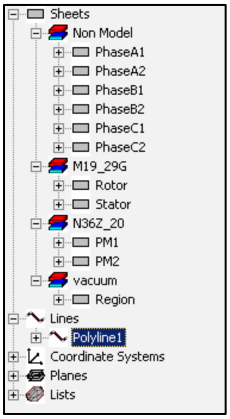


7. Name the polyline *airgap_arc* and accept the object

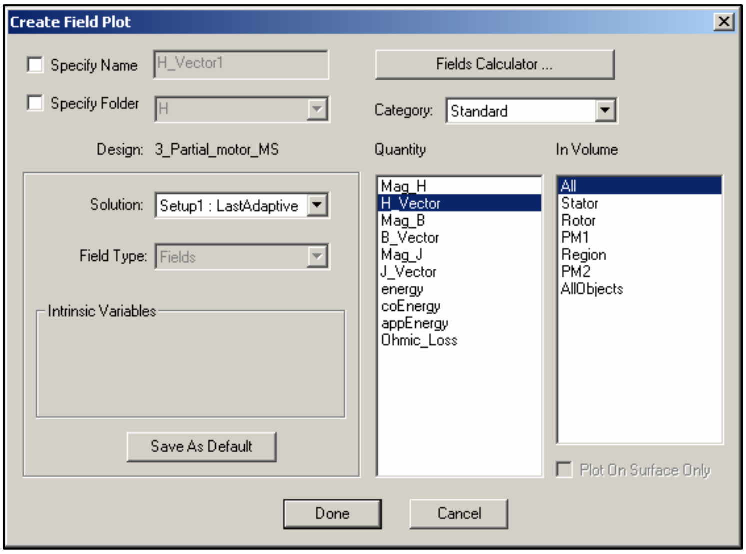


Post Processing (Cont'd)

8. A new folder 'Lines' has appeared on the object tree, containing the new defined arc.



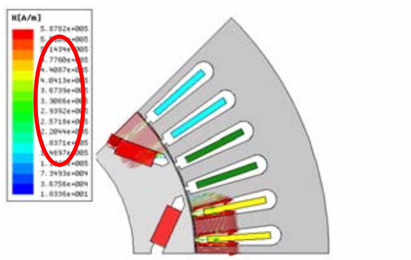
9. Select the line *airgap_arc*, move the mouse on the drawing area, right mouse click, then select the menu item *Fields > H > H_vector*.



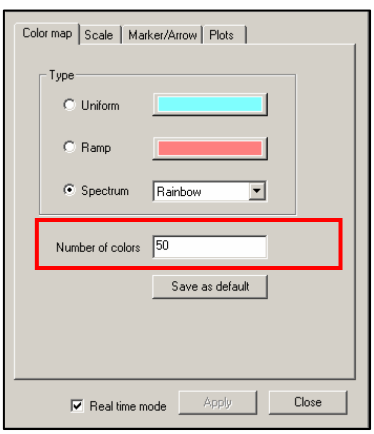
10. Accept the Field plot setting

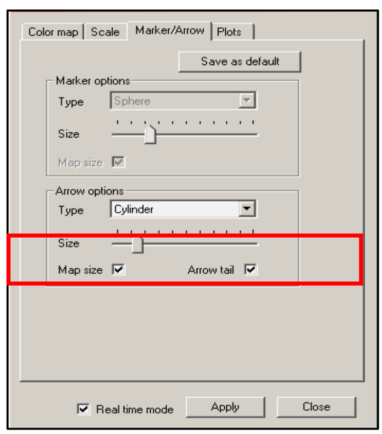
Post Processing (Cont'd)

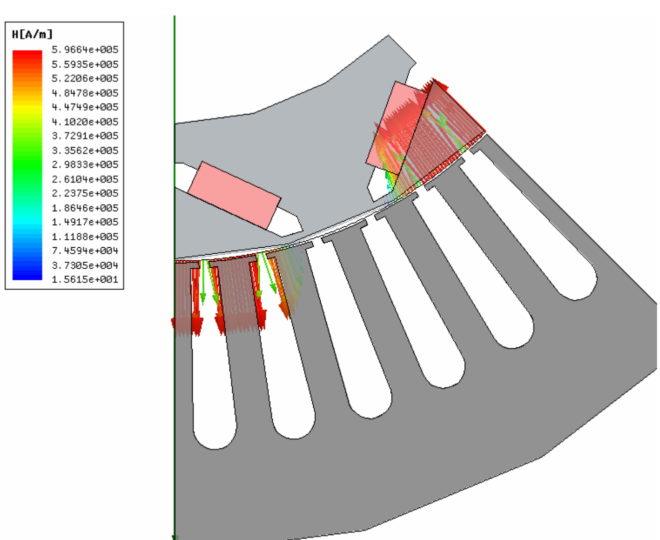
11. The vector plot of H appears with the default setting. To customize the display, double click on the scale zone:



12. You can modify the default settings in the different tabs like below:

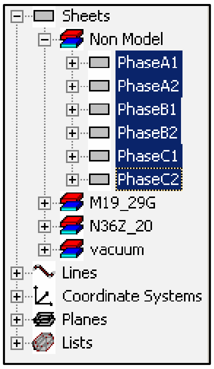


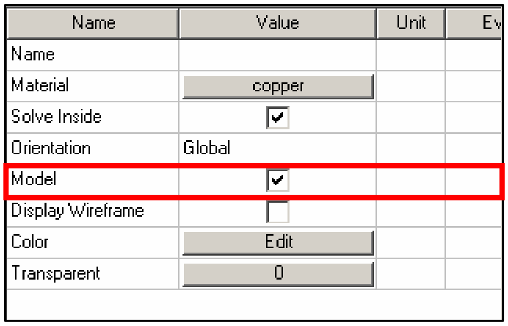




Full Load Study

- Save the project. Click on the Maxwell design '*3_Partial_motor_MS*', right mouse click and select 'Copy'.
- Click on the project name, right mouse click and select Paste. Change the copied design to '*4_Partial_motor_MS2'*.
- In this design, we apply current in the coils: we need to include the coils in the model. Select the 6 coils from the modeler tree. In the property window, select the radio button Model.





- Unhide the coils by selecting the menu item *View> Show selections> All views*

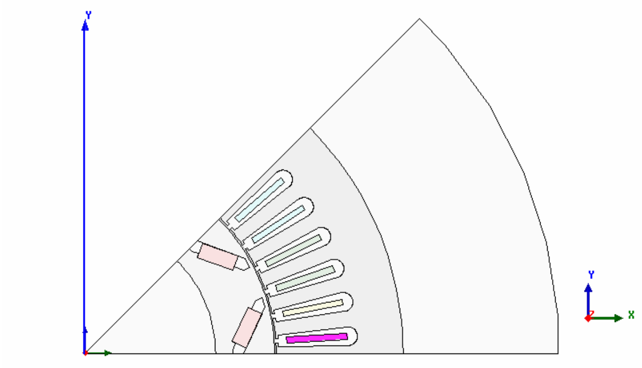
Apply Excitations

- The coils are partially represented in the model. We need to enter the current that flows in and out inside each coil. The excitation is realized through a balanced three phase system. For instance, in our example, we apply:
  - 1500 A to PhaseA
  - -750 A to PhaseB
  - -750 A to PhaseC.

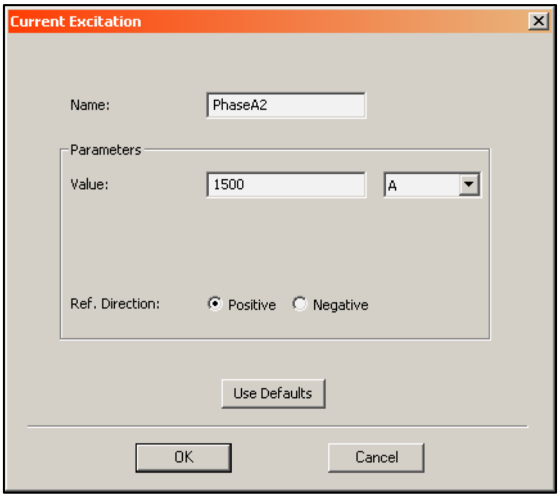
- In the Magnetosatic solver, the sources are given in terms of currents. We do not need to model each turn at this stage; therefore we only enter the total current in each phase. The number of turns and the electrical topology are only taken into account for the inductances calculation.

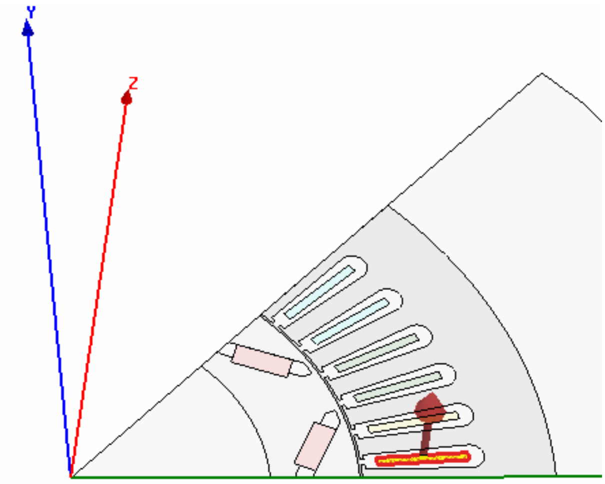
Apply Excitations (Cont'd)

- Switch the selection mode to face
- Enter Excitation for Coil *PhaseA2*:
  1. Select the *PhaseA2*



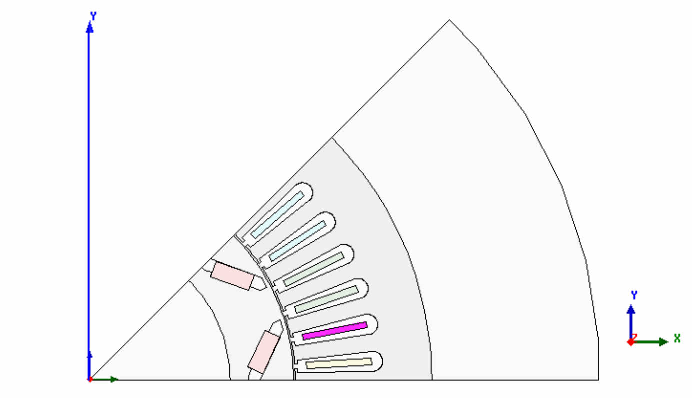
  2. Right mouse click, select the menu item *Apply Excitation > Current*
  3. Rename the Excitation *PhaseA2*
  4. Enter **1500A**
  5. As the default current direction plotted in red is good, leave **Positive**
  6. Validate the Excitation



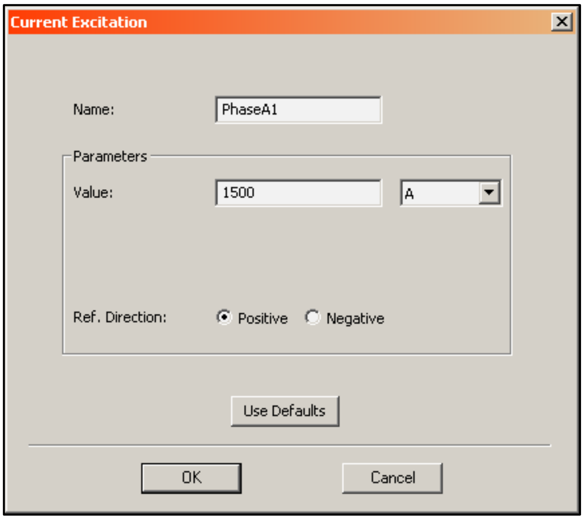


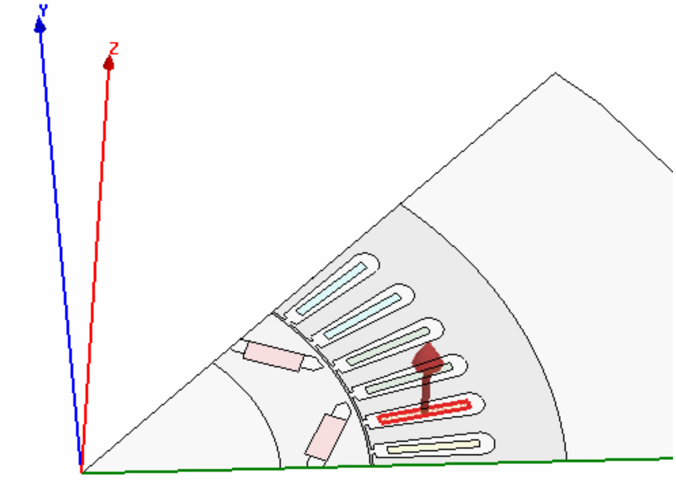
Apply Excitations (Cont'd)

- Switch the selection mode to face
- Enter Excitation for Coil *PhaseA1*
    1. Select the *PhaseA1*



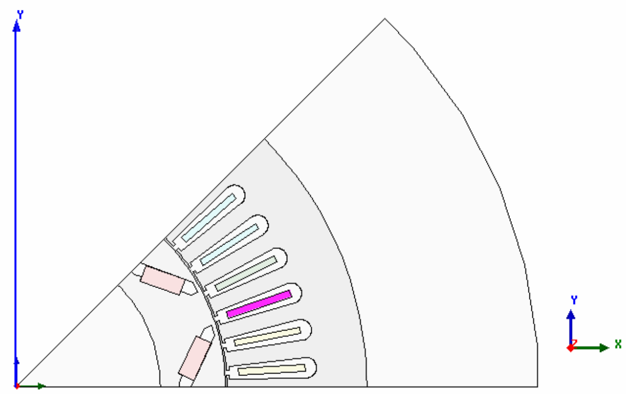
2. Right mouse click, select the menu item *Apply Excitation > Current*
3. Rename the Excitation *PhaseA1*
4. Enter **1500A**
5. As the default current direction plotted in red is good, leave **Positive**
6. Validate the Excitation



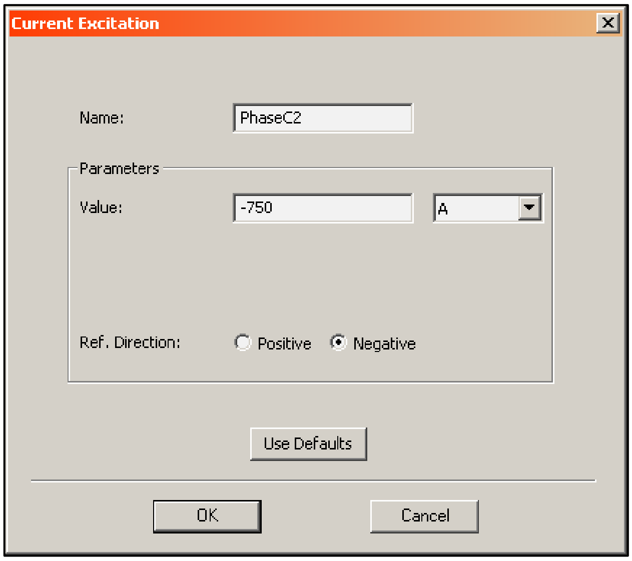


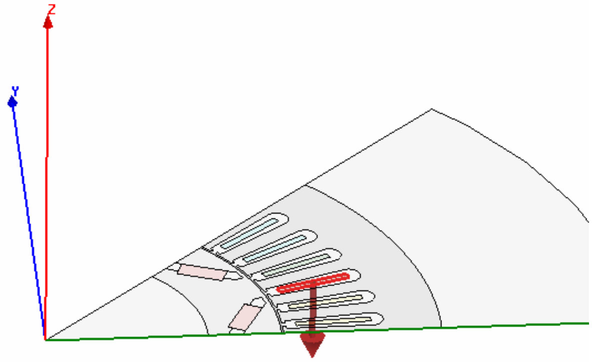
Apply Excitations (Cont'd)

- Switch the selection mode to face
- Enter Excitation for Coil *PhaseC2*
  1. Select the *PhaseC2*



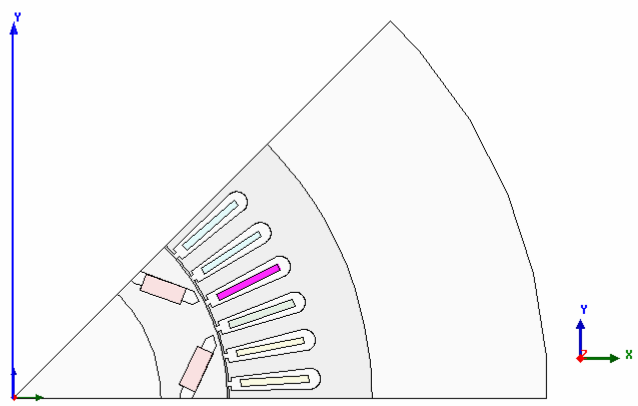
  2. Right mouse click, select the menu item *Apply Excitation > Current*
  3. Rename the Excitation *PhaseC2*
  4. Enter **-750A**
  5. As the default current direction plotted in red is not good, choose **Negative**
  6. Validate the Excitation



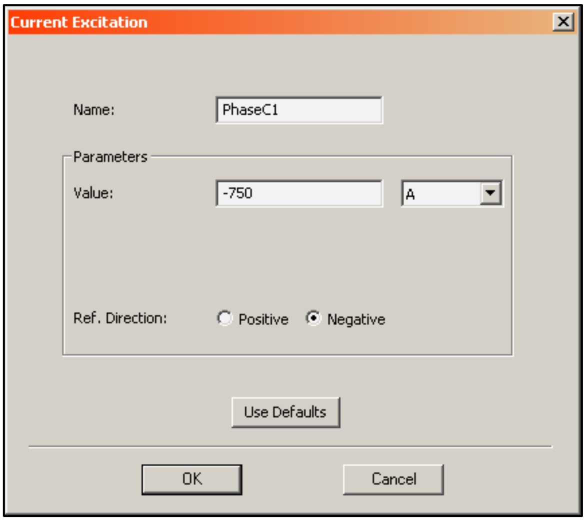


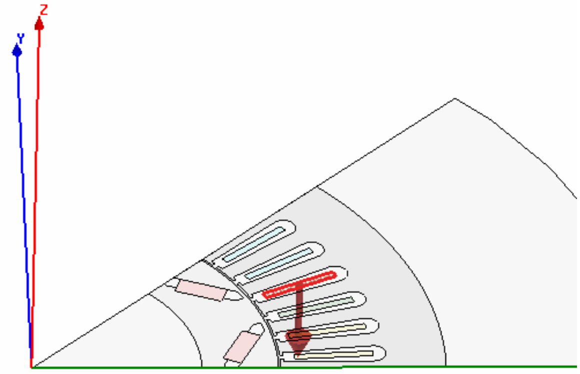
Apply Excitations (Cont'd)

- Switch the selection mode to face
- Enter Excitation for Coil *PhaseC1*
  1. Select the *PhaseC1*



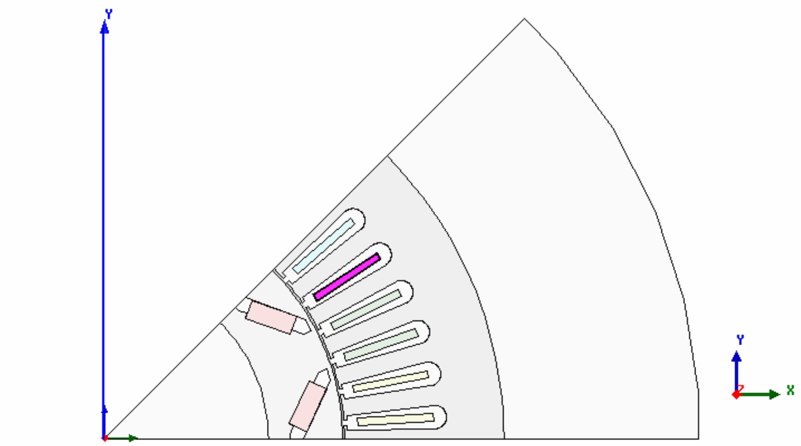
  2. Right mouse click, select the menu item *Apply Excitation > Current*
  3. Rename the Excitation *PhaseC1*
  4. Enter **-750A**
  5. As the default current direction plotted in red is not good, choose **Negative**
  6. Validate the Excitation



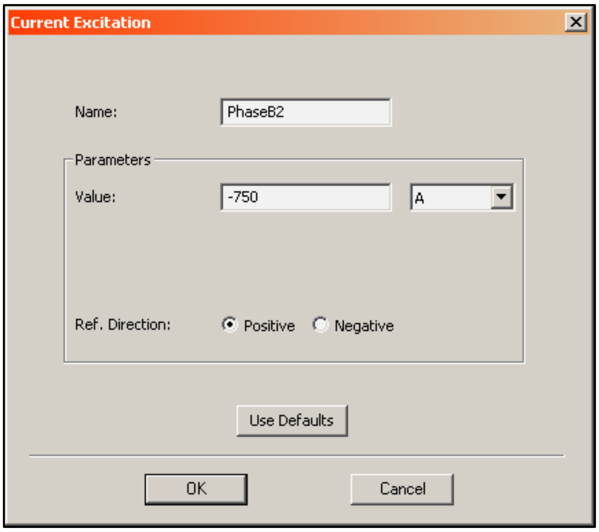


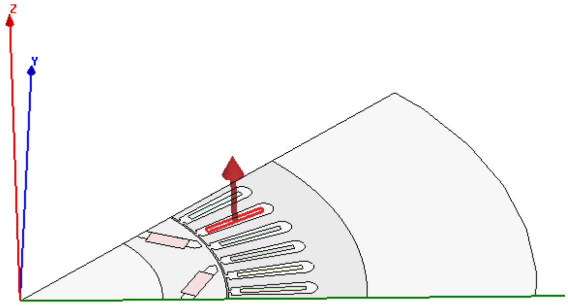
Apply Excitations (Cont'd)

- Switch the selection mode to face
- Enter Excitation for Coil *PhaseB2*
  1. Select the *PhaseB2*



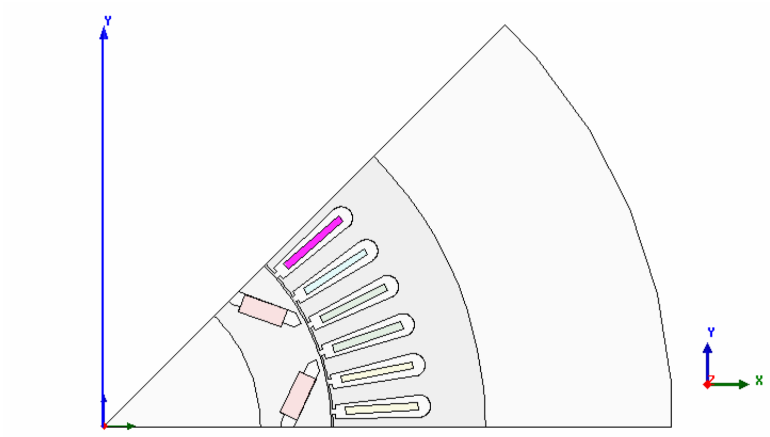
  2. Right mouse click, select the menu item *Apply Excitation > Current*
  3. Rename the Excitation *PhaseB2*
  4. Enter **-750A**
  5. As the default current direction plotted in red is good, leave **Positive**
  6. Validate the Excitation



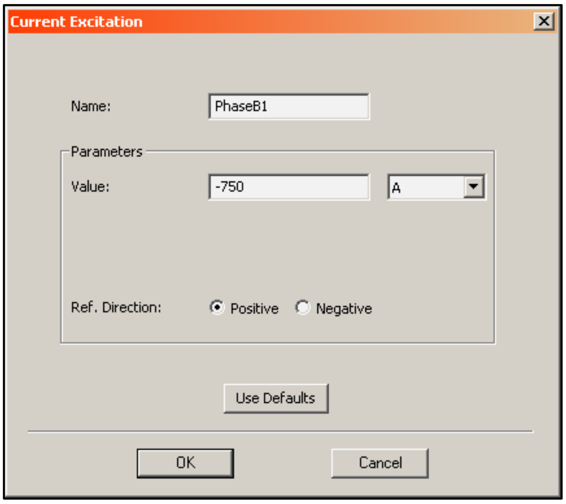


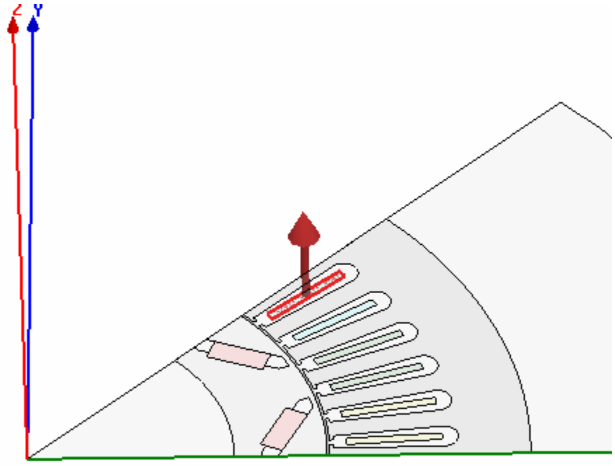
Apply Excitations (Cont'd)

- Switch the selection mode to face
- Enter Excitation for Coil *PhaseB1*
  1. Select the *PhaseB1*



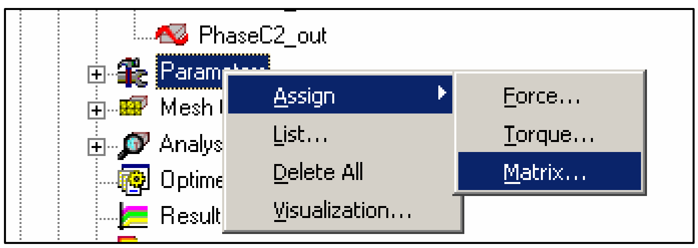
  2. Right mouse click, select the menu item *Apply Excitation > Current*
  3. Rename the Excitation *PhaseB1*
  4. Enter **-750A**
  5. As the default current direction plotted in red is good, leave **Positive**
  6. Validate the Excitation



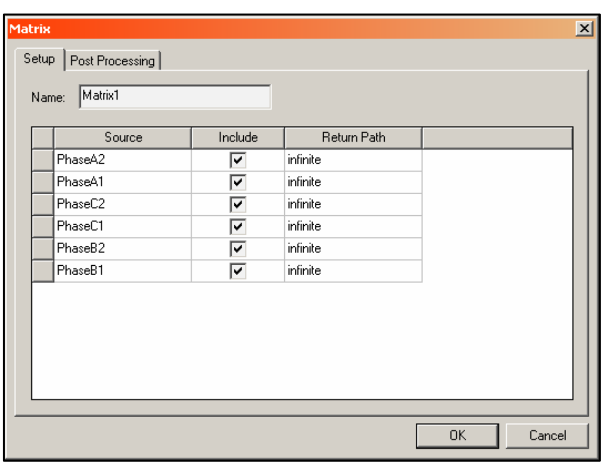


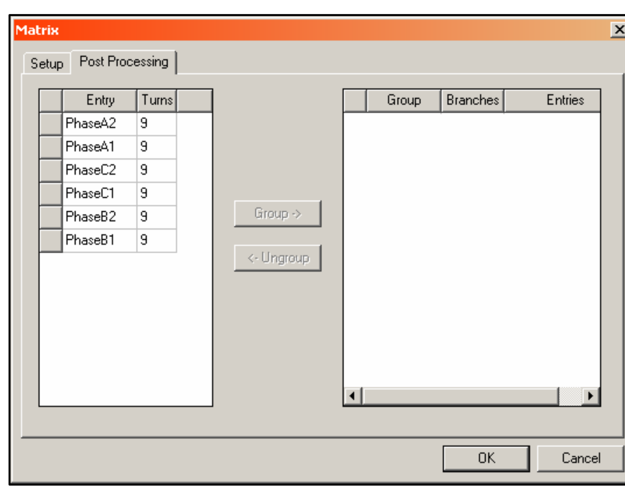
Inductance computation

- We are interested by the inductances computation. The source set up is independent from the winding arrangement: we have only entered the corresponding amp-turns for each terminal. When looking at the inductances, we obviously need to enter the number of turns for the coils and also how the coils are electrically organized.
- Select Parameters in the project tree, right mouse click and select *Assign > Matrix*



- Include the 6 phases in the matrix computation. The inductances are computed for 1 turn at this stage.

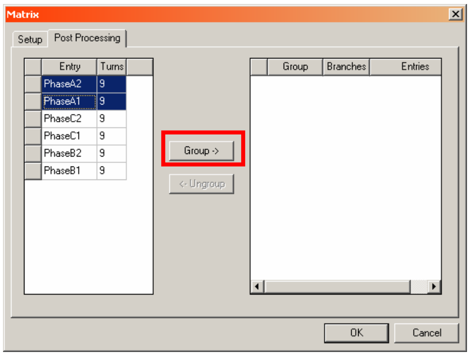


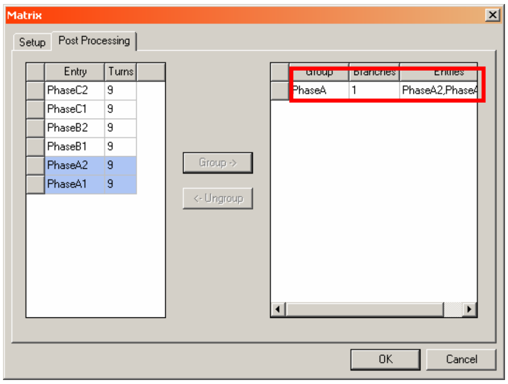


- Select the Post Processing tab. We define in this panel the number of turns for each coil. Enter 9 for the six coils.
- We also want to group all the coils of the same phase. This will enable us to have the inductance of the entire winding

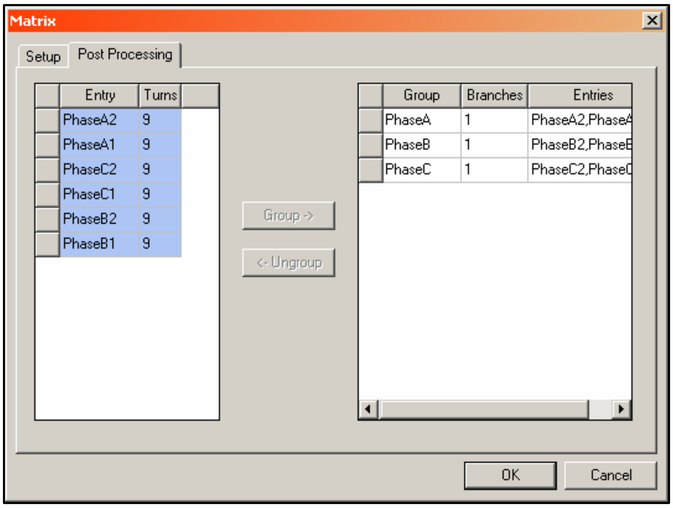
Inductance computation (Cont'd)

- Select the *PhaseA1_in* and *PhaseA2_in* entries, they hit the group button.
- Name The group *PhaseA*





- Repeat the operation for the 3 phases

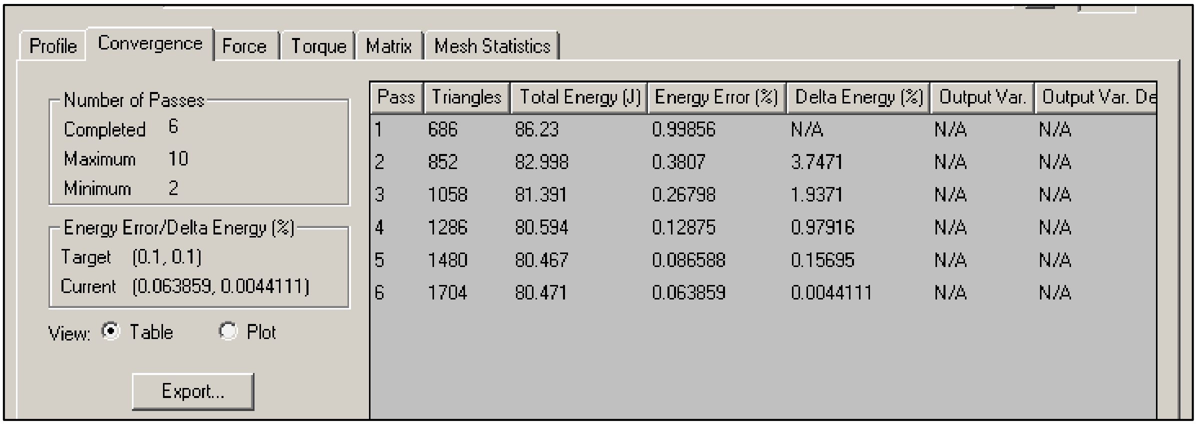


Analyse

- Right mouse click on the setup et select Analyze or click on the  icon.

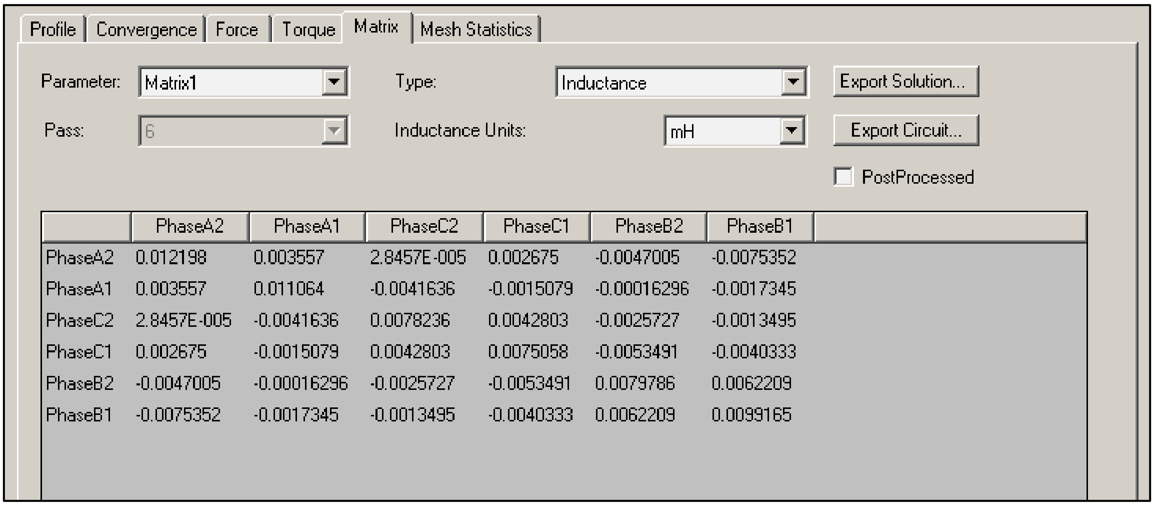
Post processing

- The computation takes 6 passes to converge. The Convergence panel can be seen by right mouse clicking on Setup1, selecting the menu item *Convergence*



Pass	Triangles	Total Energy (J)	Energy Error (%)	Delta Energy (%)	Output Var.	Output Var. De
1	686	86.23	0.99856	N/A	N/A	N/A
2	852	82.998	0.3807	3.7471	N/A	N/A
3	1058	81.391	0.26798	1.9371	N/A	N/A
4	1286	80.594	0.12875	0.97916	N/A	N/A
5	1480	80.467	0.086588	0.15695	N/A	N/A
6	1704	80.471	0.063859	0.0044111	N/A	N/A

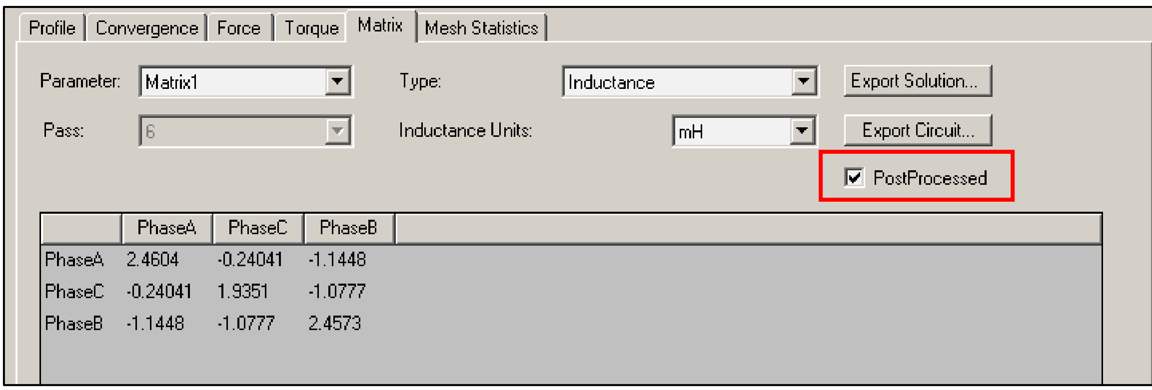
- Inductance values. Select the ‘*Solutions’* tab. The inductance for each coils appears. It is assumed that each coil has only one turn.



	PhaseA2	PhaseA1	PhaseC2	PhaseC1	PhaseB2	PhaseB1
PhaseA2	0.012198	0.003557	2.8457E-005	0.002675	-0.0047005	-0.0075352
PhaseA1	0.003557	0.011064	-0.0041636	-0.0015079	-0.00016296	-0.0017345
PhaseC2	2.8457E-005	-0.0041636	0.0078236	0.0042803	-0.0025727	-0.0013495
PhaseC1	0.002675	-0.0015079	0.0042803	0.0075058	-0.0053491	-0.0040333
PhaseB2	-0.0047005	-0.00016296	-0.0025727	-0.0053491	0.0079786	0.0062209
PhaseB1	-0.0075352	-0.0017345	-0.0013495	-0.0040333	0.0062209	0.0099165

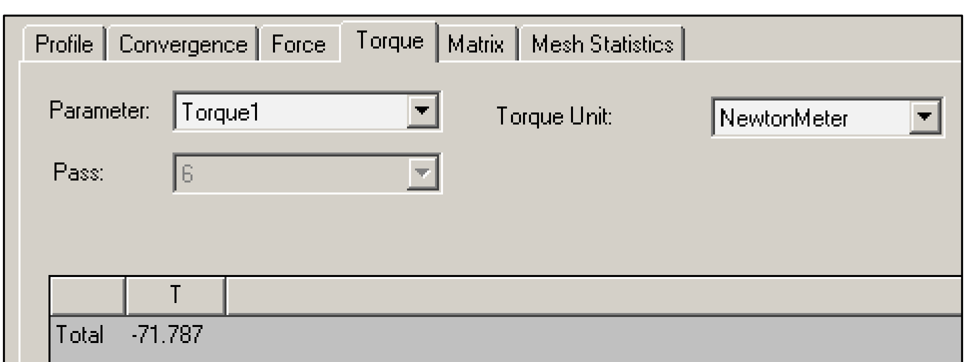
Post processing (Cont'd)

- Select the radio button *Post Processed*. The inductance for each winding is displayed



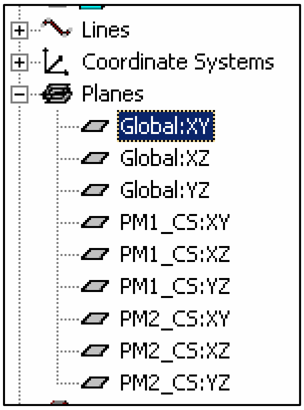
- **Note:** it is possible to export the inductance matrix to Simplorer using the *Export Circuit* button

- **Torque value.** Select the '*Solutions*' tab, Select from the pull down menu Torque1. The torque for the full motor needs to be multiplied by 8 (symmetry factor), then multiplied by 0.083 (length of the motor). This gives around 47N.m. In this case, we have not synchronized the position of the rotor poles with the winding currents, so we are far from the optimized excitation value to obtain a maximum torque. Different angles between the rotor and the stator would give different values.

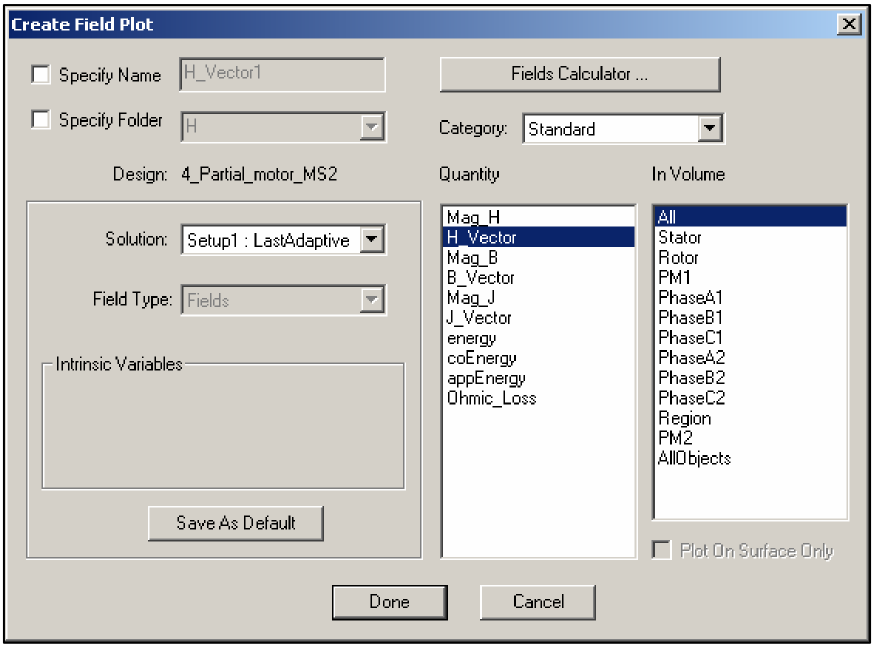


Post processing (Cont'd)

- **Plot the H field on the plane XY.** Select the plane XY belonging to the global Coordinate System in the modeler tree



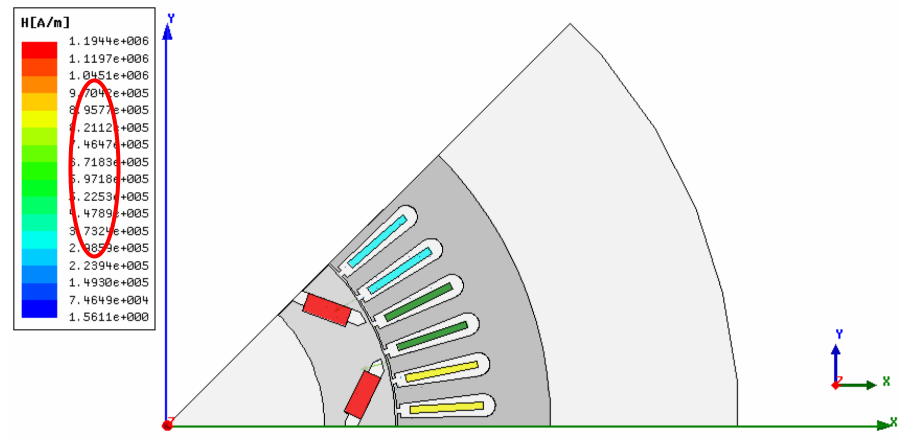
- Move the mouse pointer to the drawing area, right mouse click and select the menu item *Fields > H > H_vector*



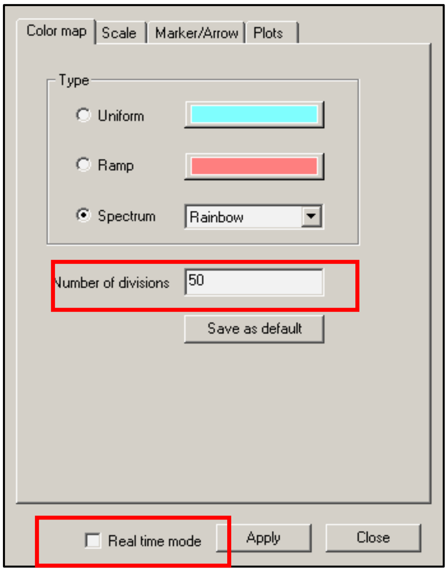
- Validate the setting

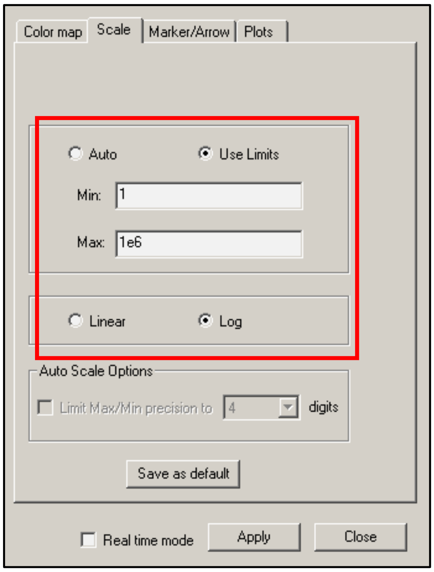
Post processing (Cont'd)

- With the default parameters, the H vectors are too small. Double click on the scale zone



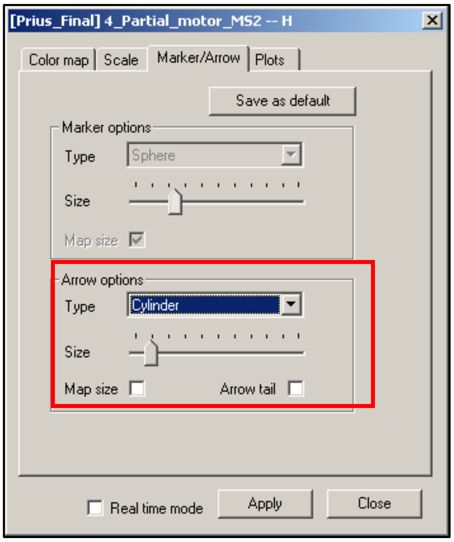
- On the *'Color Map'* tab, uncheck the '***Real time mode***' button and change the number of colors to 50
- On the *'Scale tab'*, Check the ***Use Limits*** button, then Enter **1** and **1e6** for the limits. Also, Check the ***Log*** button to have a log scale.

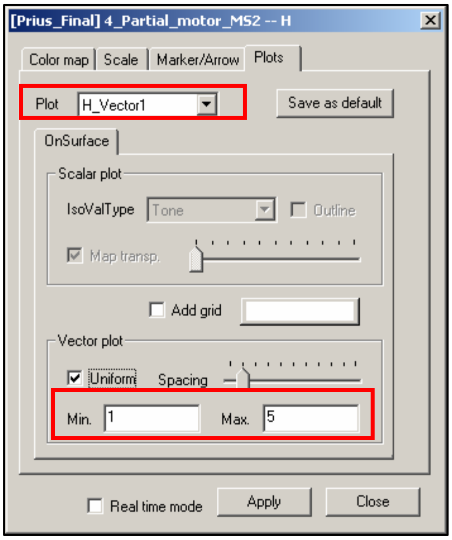




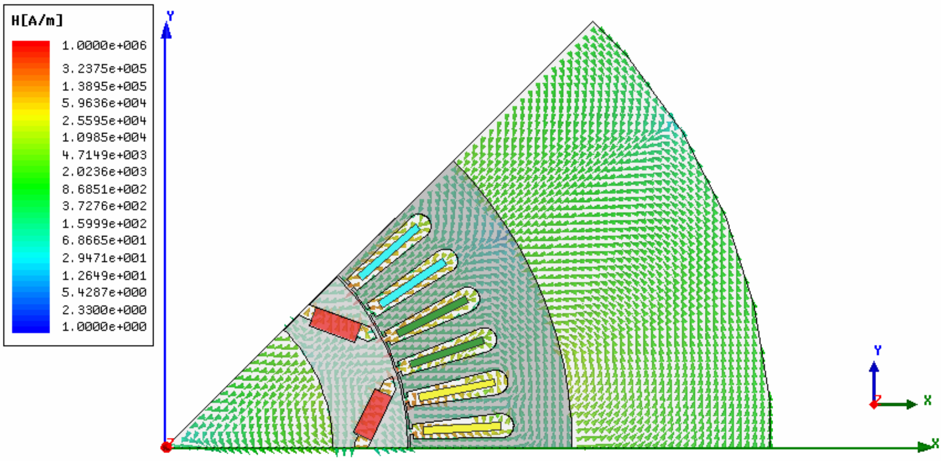
Post Processing (Cont'd)

- On the '*Marker/Arrow*' tab, reduce the size of the arrow, then uncheck the ***Mapsize*** and ***Arrow tail*** buttons.
- On the '*Plots*' tab, make sure the right plot context is selected, then modify the Vector plot min and max to 1 and 5



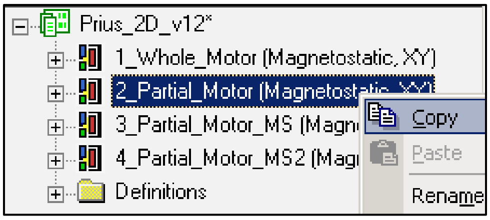


- We obtain the following plot. The H field is stronger around phase A as the input current is higher.



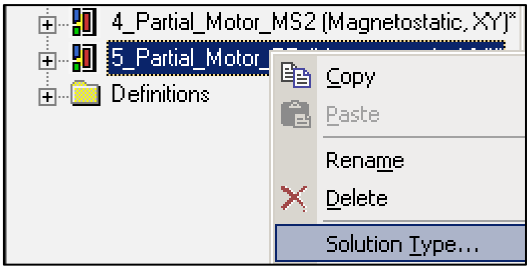
DYNAMIC ANALYSIS

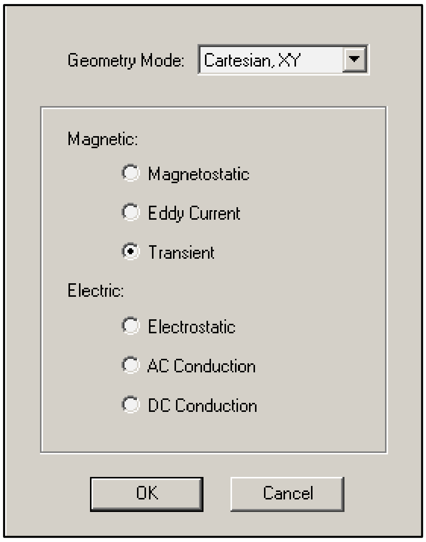
- We will study the transient characteristic of the motor.
- Save the project. Click on the Maxwell design '*2_Partial_motor*', right mouse click and select 'Copy'.



- Click on the project name, right mouse click and select Paste. Change the copied design to '*5_Partial_motor_TR*'.

- Select the design name from the project manager, Right mouse click and change the solution type from *Magnetostatic* to *Transient*.

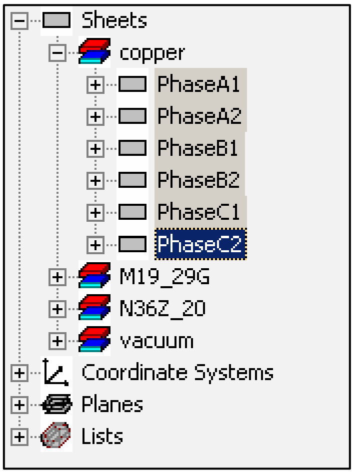


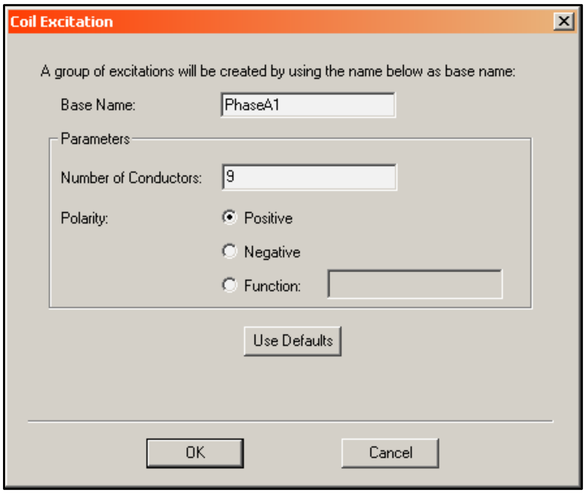


- The transient solver acts differently from the Magnetostatic solver mainly because:
  - There is not adaptive meshing. Since the geometry changes at every time step, Maxwell does not re-mesh at every time step adaptively for obvious time reason. In transient analysis, we will build a good mesh valid for all the rotor positions.
  - The sources definition is different. In Magnetostatic, we were only interested in the total current flowing into conductor. In Transient, we use stranded conductors (the exact number of conductors is required for each winding) as the current can be an arbitrary time function. We need to create dedicated coils and windings.

Create Coils

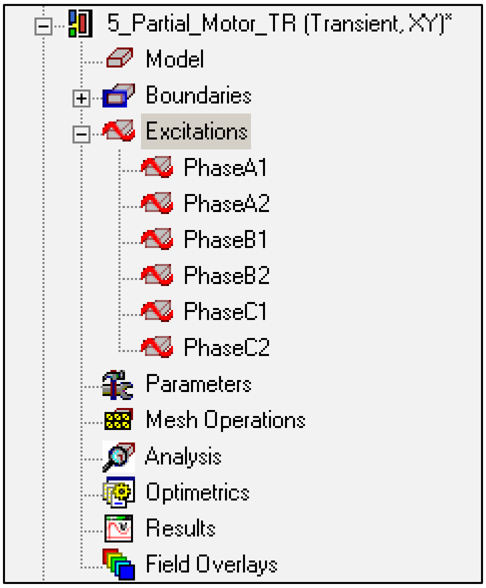
- Select the 6 coils PhaseA1, PhaseA2, PhaseB1, PhaseB2, PhaseC1 and PhaseC2.
- Right mouse click and select the menu item *Assign Excitation > Coil*
  1. Leave the default name as it will be automatically affected using the object's name
  2. Enter 9 for the number of conductors





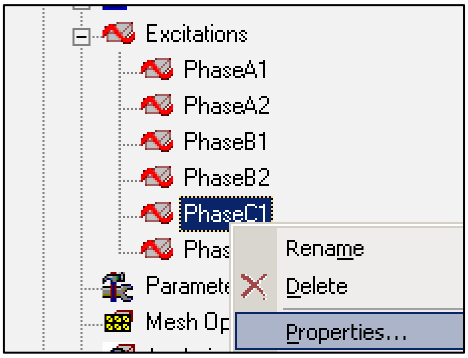
Create Coils (Cont'd)

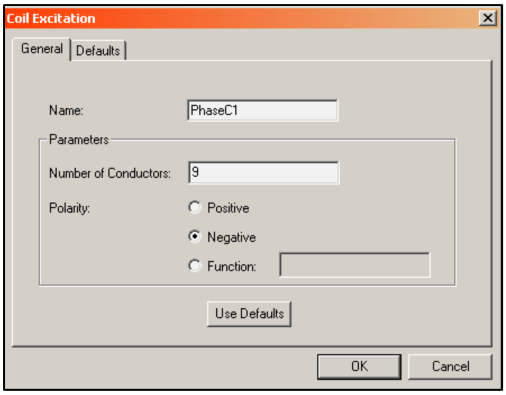
- The six coils definitions are processed.



- We need to change the orientation of the coils for **PhaseC1** and **PhaseC2**:

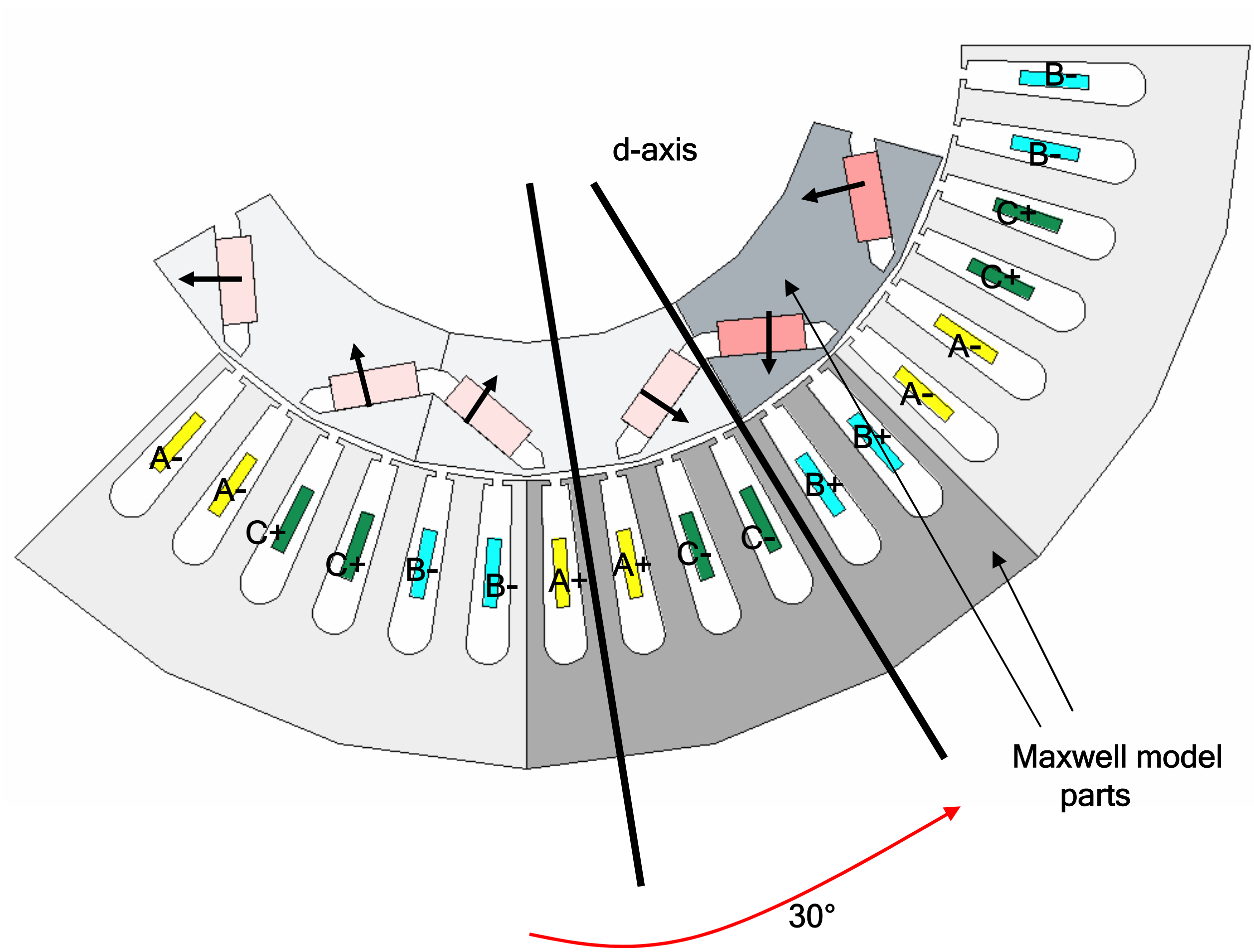
1. Select **PhaseC1**, in the Project Manager Tree
2. Right mouse click and select Properties
3. Switch the polarity from **Positive** to **Negative**
4. Repeat 1-3 with Coil **PhaseC2**





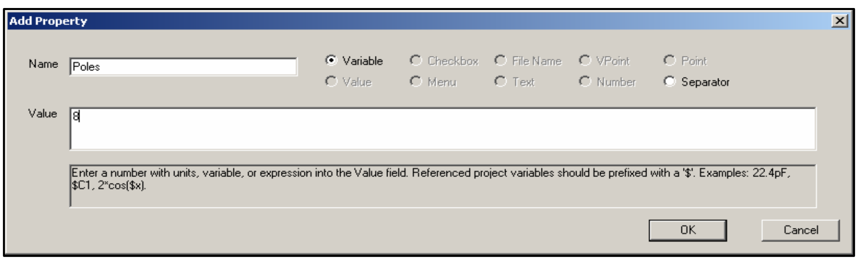
Motor excitation

- The IPM motor is such that the rotor is in synchronism with the phase excitation. The excitation is such that the flux due to the permanent magnet is maximized in synchronization with the rotor movement.
- The excitation is a 3 phase balanced current. The phase sequence is A+C-B+
- At t=0, the A-phase has to be in the opposite axis to the d-axis. Therefore we have to move the initial position of the rotor by 30 deg such that the pole be aligned at the middle of A+A-

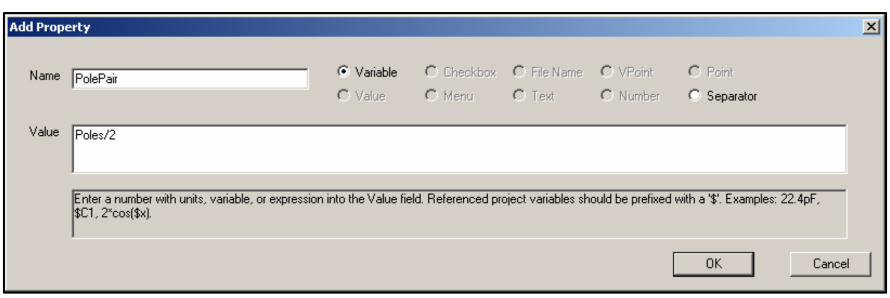


Create Parameters for excitations

- We need to define parameters that will be used to define the excitation
- Select the menu item *Maxwell 2D> Design Properties*
- The parameters window appears
- Click on the Add button to add the number of poles of the motor
  - Enter **Poles** in the name area
  - **8** in the value area
  - Click on OK to accept the parameter



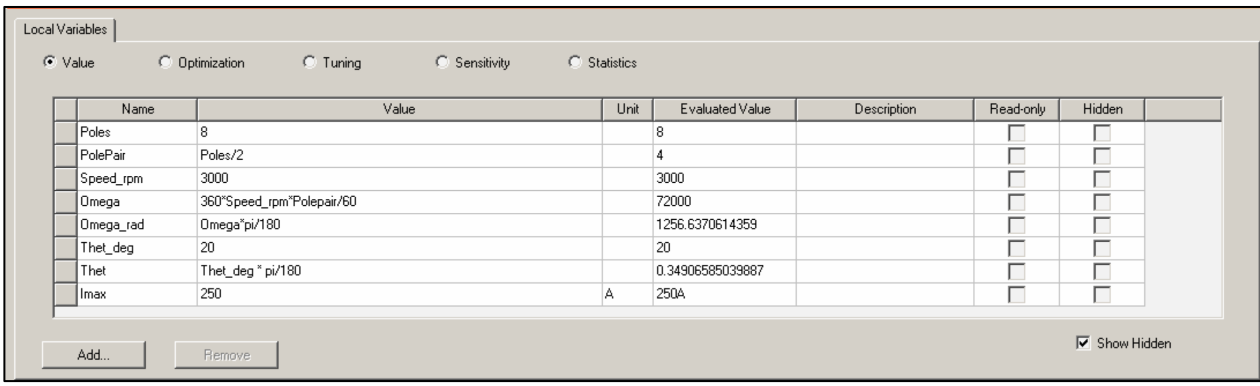
- Using the same method enter:
  - *PolePair*, the number of pair of poles; its value is *Poles/2*



  - *Speed_rpm*, the speed of the motor in rpm; its value is *3000*
  - *Omega*, the pulsation of the excitation in degrees/s; its value is *360*Speed_rpm*Polepair/60*
  - *Omega_rad* the pulsation in rad/s; its value is *Omega * pi / 180*
  - *Thet_deg* the load angle of the motor ; for instance, we use 20 degrees in this study; enter *20 deg.*
  - *Thet* is load angle in radian therefore its value is *Thet_deg * pi /180*
  - *Imax* the peak winding current of the motor; its value is **250A.**

Create Parameters for excitations (Cont'd)

- The design properties panel will eventually look like:



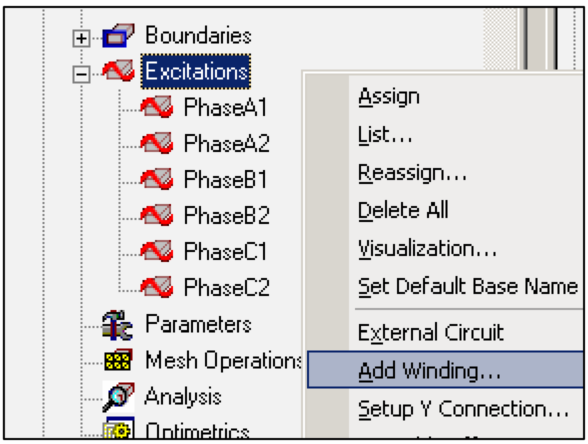
Name	Value	Unit	Evaluated Value	Description	Read-only	Hidden
Poles	8		8			
PolePair	Poles/2		4			
Speed_rpm	3000		3000			
Omega	360*Speed_rpm*Polepair/60		72000			
Omega_rad	Omega*pi/180		1256.6370614359			
Thet_deg	20		20			
Thet	Thet_deg * pi/180		0.34906585039887			
Imax	250	A	250A			

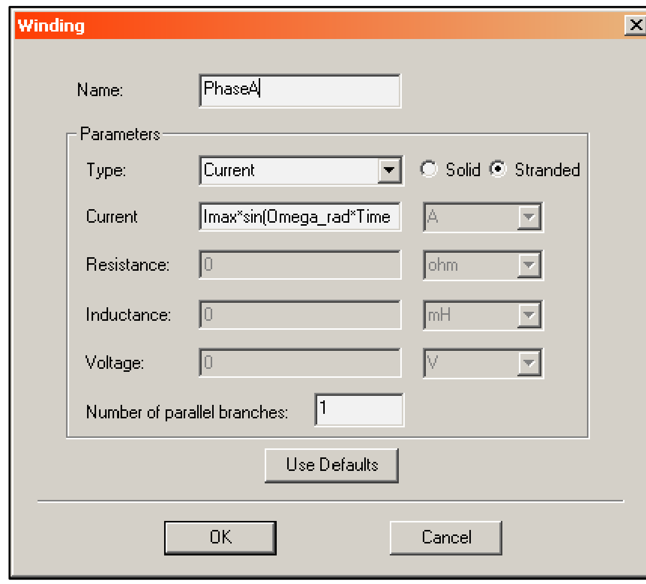
Create Windings

- The terminals are meant to define the excitation paths in and out of the model. The actual excitation is defined through the definition of windings. A winding needs to be defined for each parallel electrical excitation of the motor.
- The motor is excited with a balanced three phase connection. A sinusoidal excitation is applied. At each time step, the phases have a 120 degree shift. The load angle is also added.

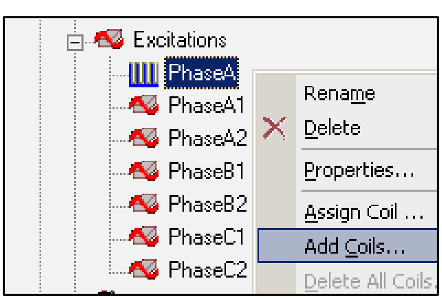
Create Windings (Cont'd)

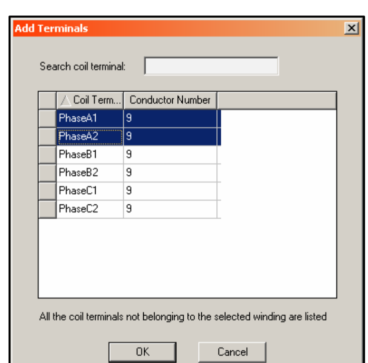
- **Winding PhaseA**.. From the project tree, right mouse click on *Excitations*, then select the menu item *Add Winding*





1. Enter *PhaseA* for the name.
2. Select Stranded because each terminal has 9 turns
3. Enter winding current: *Imax*sin(Omega_rad*Time+Thet). Time* is the internal reserved variable for the current time.
4. Click on OK
5. Right mouse click on the winding *PhaseA* from the project tree, select the menu item *Add Coils*





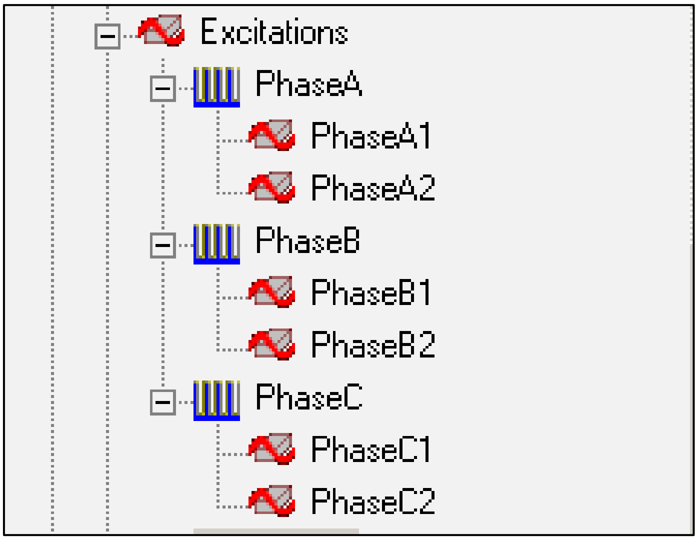
6. Select the 2 *PhaseA* coils (using the Ctrl button) and click on OK

Create Windings (Cont'd)

- **Winding PhaseB**.. From the project tree, right mouse click on *Excitations*, then select the menu item *Add Winding.* Repeat the same operation using :
  - Name the Winding *PhaseB*
  - The winding current is *Imax*sin(Omega_rad*Time-2*pi/3+Thet).* It is shift by -120 degrees from PhaseA.
  - Select the 2 *PhaseB* t coils

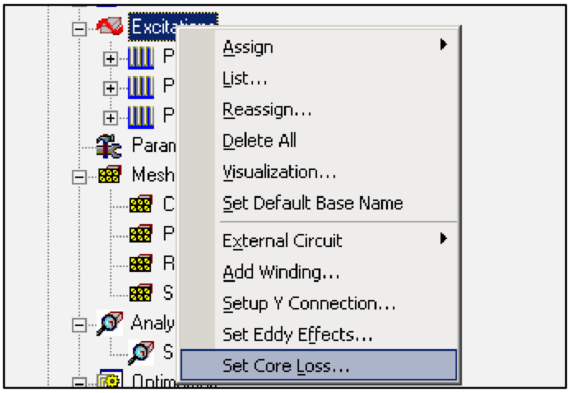
- **Winding PhaseC**.. From the project tree, right mouse click on *Excitations*, then select the menu item *Add Winding.* Repeat the same operation using :
  - Name the Winding *PhaseC*
  - The winding current is *Imax*sin(Omega_rad*Time+2*pi/3+Thet).* It is shift by +120 degrees from PhaseA.
  - Select the 2 *PhaseC* coils

- The project tree should now have the terminals sorted under each Winding:

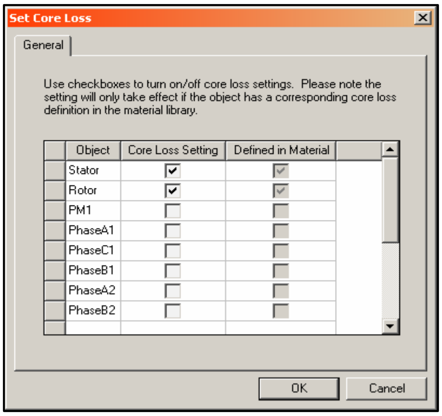


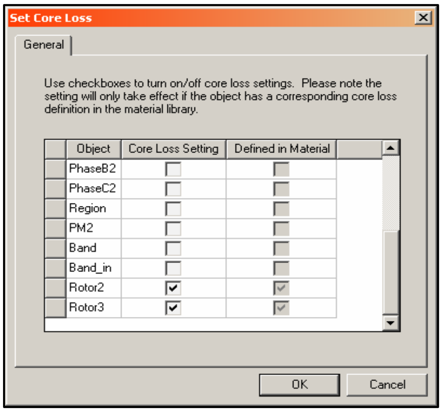
Add Coreloss computation

- The coreloss are not activated by default. If you wish to have them considered, expand the project tree window, right mouse click on *Excitations> Set Core Loss*



- Select the steel objects: Stator, Rotor, Rotor2 and Rotor3
- Accept the Setting



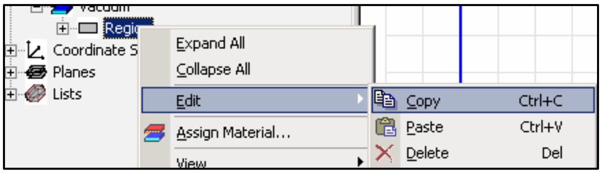


- **Note**: you need to have the coreloss parameters defined in the material setup

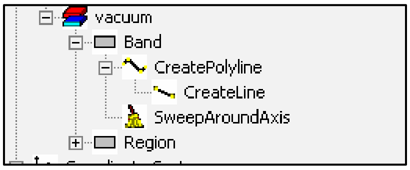
Add Band object

- The moving parts (rotor and permanent magnets) need to be enclosed in an air object, the band. This will separated the moving part from the fixed part of the project. Some rules apply for the definition of the band object for motor applications:
  - The band object must be somewhat larger than the rotating parts in all directions (except at the boundaries)
  - The band object should be a facetted type cylinder of wedge
  - It is very advisable to have an air object that encloses all the moving object inside the band object. This will facilitate the mesh handling around the air gap

- To create the Band object, we will clone the region and adapt the parameters:
  1. Select the object Region, Right Mouse click, then Select Edit > Copy



  2. Use the *Ctrl+V* key combinaison to paste the *Region*.
  3. Change the name of the object from *Region1* to *Band*
  4. Expand the history tree of the Object



Add Band object (Cont'd)

5. Double click on the *CreateLine* command. The rotor radius is 80.2mm. The inner diameter of the stator 80.95mm. We pick the middle for Band object . Enter *80.575,0,0* mm instead of 200,0,0 for *Point2*

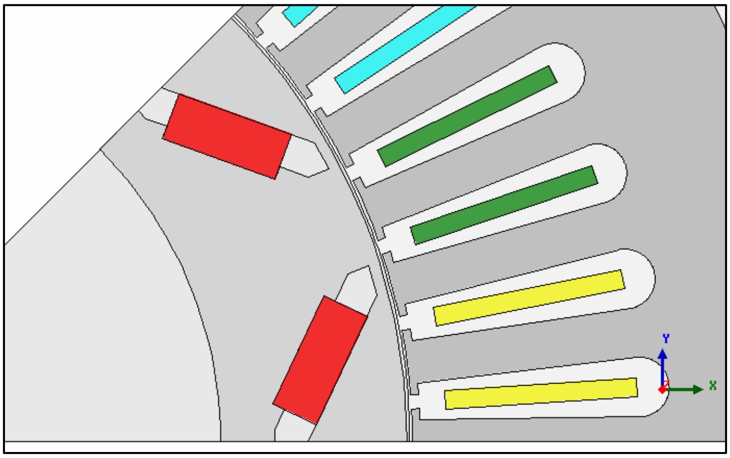
Name	Value	Unit
Segment Type	Line	
Point1	0 ,0 ,0	mm
Point2	80.575 ,0 ,0	mm

6. Double Click on the *Sweep AroundAxis* command.
7. Change the *Number of Segments* from **5** to **45** so that each segment of the line covers one degree

Command

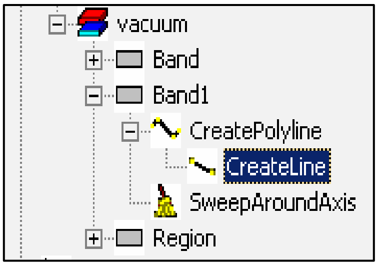
Name	Value	Unit	Evaluated Value
Command	SweepAroundAxis		
Coordinate System	Global		
Axis	Z		
Angle	45	deg	45deg
Draft angle	0	deg	0deg
Draft type	Round		
Number of Segments	45		45

8. Leave the material to *Vacuum*.



Add Band object (Cont'd)

- We now create an object that enclosed the moving objects inside the *Band*. Select the *Band* object, right mouse click, the select the menu item *Edit > Copy* or use *Ctrl-C*.
- Paste another copy of the Band object by right mouse clicking and selecting *Edit > Paste* or with the *Ctrl-V*. A new object *Band1* has been added to the object list. Expand its history tree, then double click on the *CreateLine* command



	Name	Value	Unit
	Segment Type	Line	
	Point1	0 ,0 ,0	mm
	Point2	80.4 ,0 ,0	mm

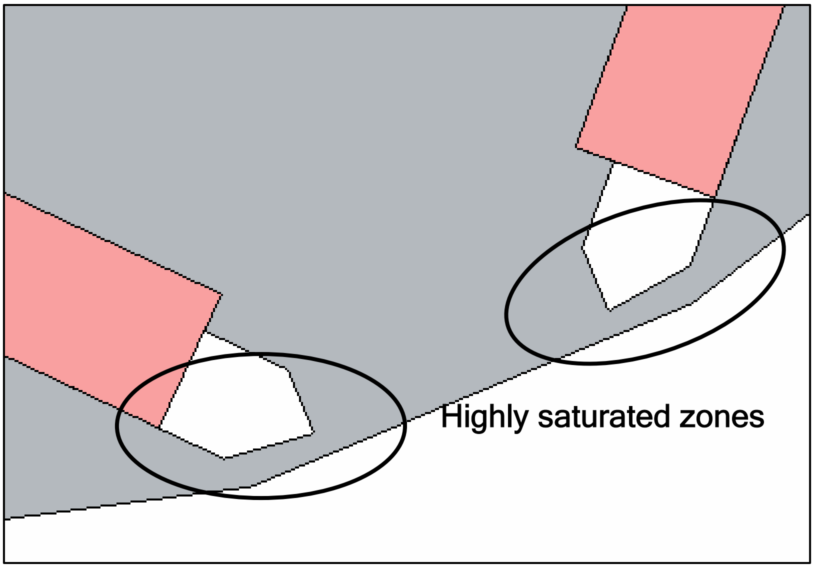
- Edit the Point2: Enter *80.4, 0, 0* mm
- This operation resizes the object to strictly cover the rotor and the permanent magnets
- Rename the *Band1* object to *Band_in*

- **Note**: We will assign the motion after the mesh operations because we will have to add objects dedicated to the meshing in the moving part

Mesh Operations

- The transient solver does not use adaptive meshing because this would require to refine the mesh at every time steps, leading to very high computation time. Using Mesh operations, we will define a decent mesh for the full transient simulation.

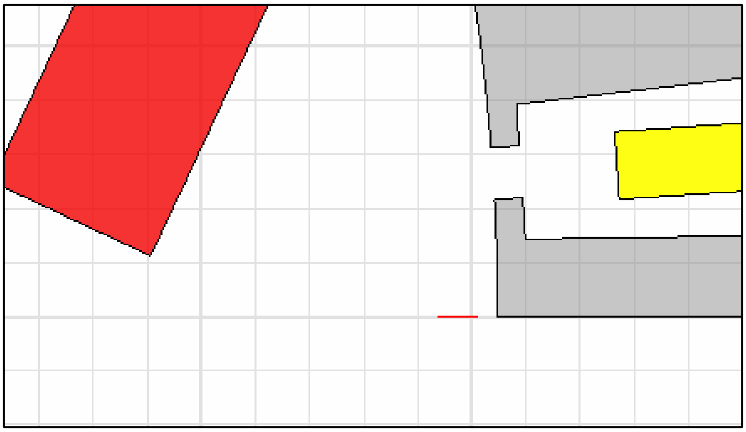
- The Rotor is designed to be highly saturated around the permanent magnets, close to the air gap. It is required to have a good mesh density around this area.



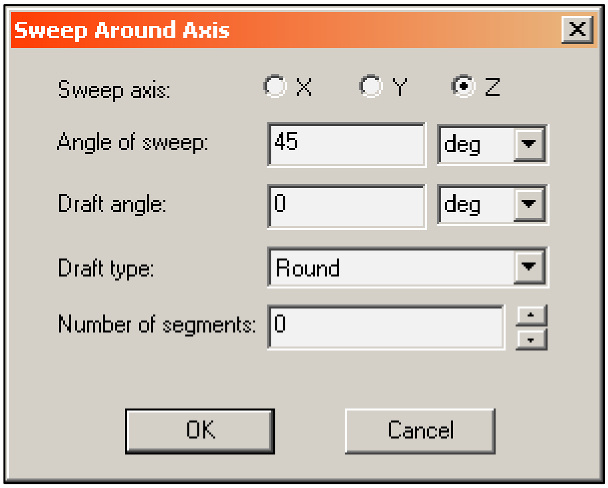
- To achieve this requirement, we create a couple of objects inside the rotor, then mesh operations will be applied to these objects in order to have a nice mesh around the ducts.
- Select the menu item *Draw > Line* or select the icon from the toolbar.
  1. Enter *78.72,0 ,0* mm for the position of Point1 and hit Enter
  2. Enter *80.2,0 ,0* mm for the position of Point2 and hit Enter twice
  3. Name the line *Rotor2*

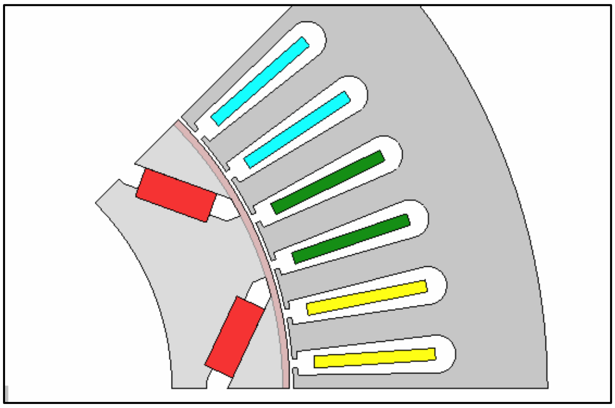
Mesh Operations (Cont'd)

- The line looks like below:



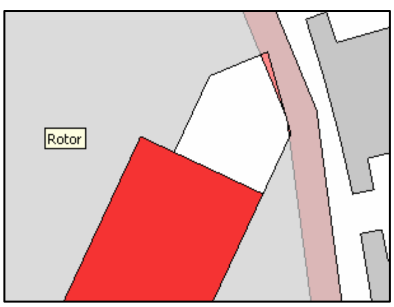
- Select the *Rotor2* object, right mouse click and select the menu item *Edit > Sweep > Around Axis.*
  - Enter the parameters as below. Note that The *Rotor* object has been created with an UDP which produces true surface, therefore our mesh object *Rotor2* has to have true surfaces. As a consequence, we enter **0** for the number of segments.

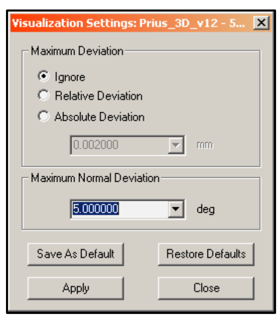


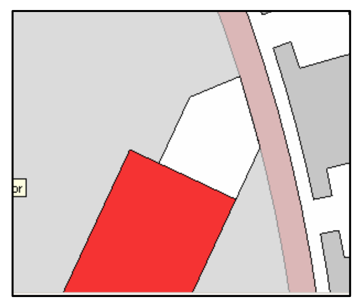


Mesh Operations (Cont'd)

- Change the material property of *Rotor2* to *M19_29G*. Also, assign some color and transparency.
- **Note**: since *Rotor2* is entirely inside *Rotor*, we do not need to apply Boolean operations.
- **Note**: because of the finite number of pixels on the computer's screen, true surfaces are represented as facetted surfaces. Also, for the same reason, the object *Rotor2 seems* to intersect with the ducts but this is not the case. You can modify the default visualization setting using: *View > Visualization Setting*







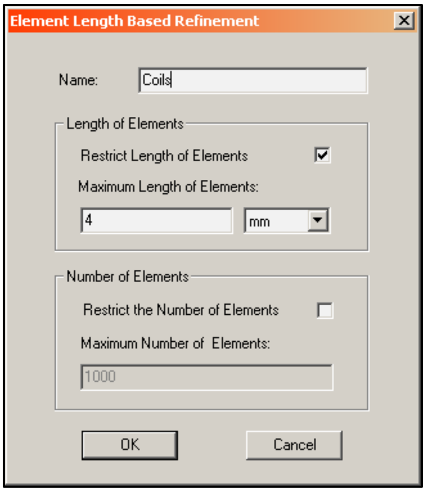
- Repeat the same operation to create the object *Rotor3*:
  1. Draw a line with dimensions:

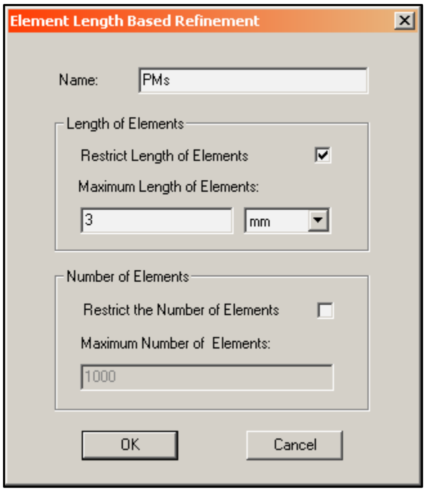
	Name	Value	Unit
	Segment Type	Line	
	Point1	78.72 ,0 ,0	mm
	Point2	79.46 ,0 ,0	mm

  2. Sweep the rectangle around Z axis
  3. Change the material property to M19_29G

Mesh Operations (Cont'd)

- Select the six coils *PhaseA1, PhaseA2*, *PhaseB1,PhaseB2*, PhaseC1 and PhaseC2. Right mouse click, select *Assign Mesh Operations > Inside Selection > Length Based*.
  1. Name the operation *Coils*
  2. Check the button Restrict Length of Elements
  3. Enter *4mm*
  4. Uncheck the button Restrict the Number of Elements
  5. Validate

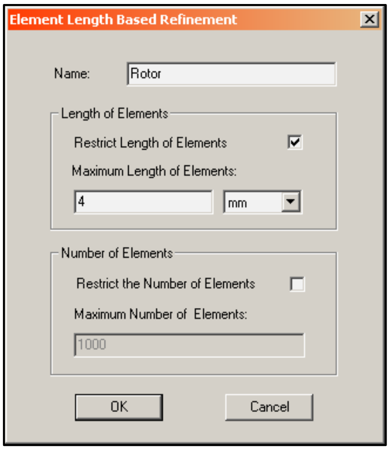


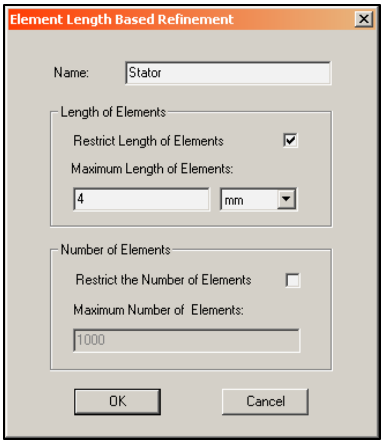


- Select the permanent magnets *PM1* and *PM2*. Right mouse click, select *Assign Mesh Operations > Inside Selection > Length Based*.
  1. Name the operation *PMs*
  2. Check the button Restrict Length of Elements
  3. Enter *3mm*
  4. Uncheck the button Restrict the Number of Elements
  5. Validate

Mesh Operations (Cont'd)

- Select the *Rotor*. Right mouse click, select *Assign Mesh Operations > Inside Selection > Length Based*.
  1. Name the operation *Rotor*
  2. Check the button Restrict Length of Elements
  3. Enter *4mm*
  4. Uncheck the button Restrict the Number of Elements
  5. Validate

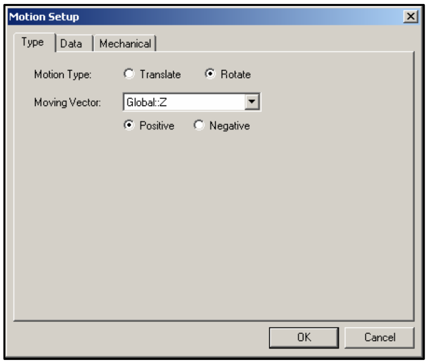


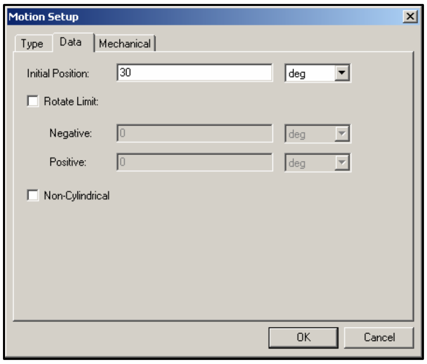


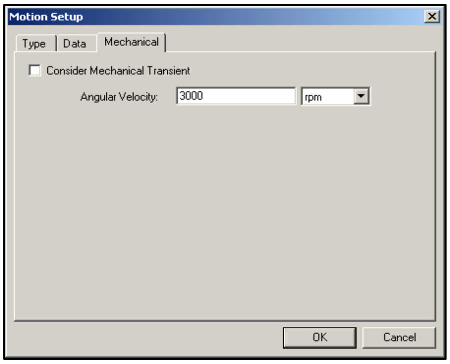
- Select the *Stator*. Right mouse click, select *Assign Mesh Operations > Inside Selection > Length Based*.
  1. Name the operation *Staor*
  2. Check the button Restrict Length of Elements
  3. Enter *4mm*
  4. Uncheck the button Restrict the Number of Elements
  5. Validate

Assign Movement

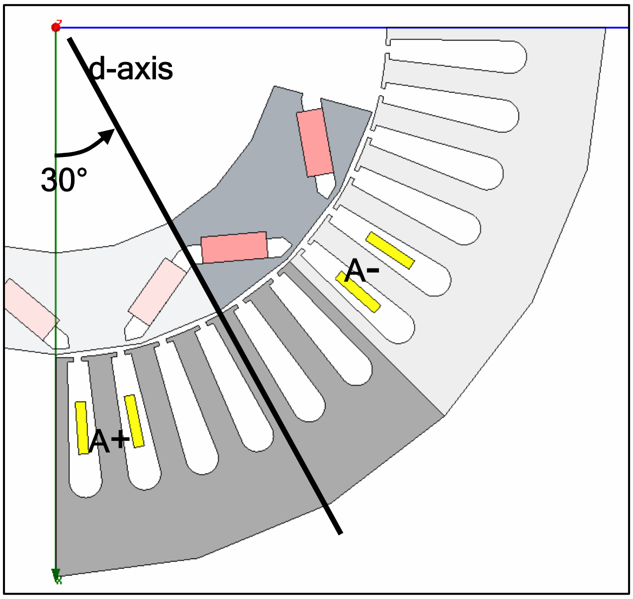
- Select the *Band* object, right mouse click and select the menu item Assign Band







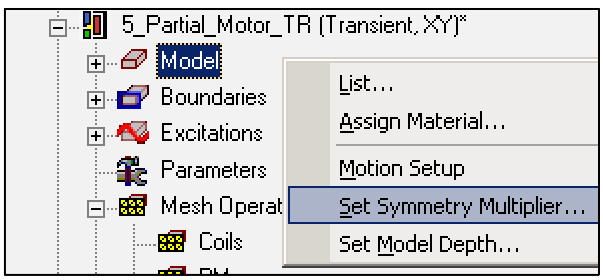
- In the *Type* tab:
  - check the *Rotate* motion button
  - Make sure that the *Global:Z* axis is selected
  - Select the *Positive* direction
- In the *Data* tab:
  - Enter *30 deg* for the initial position. The initial position of this synchronous motor is such that the A phase is opposite to the d-axis.



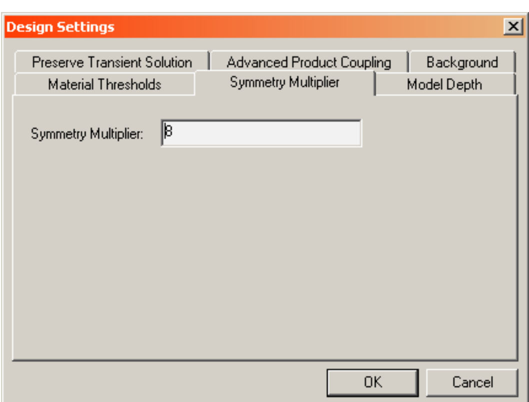
Assign Movement (Cont’d)

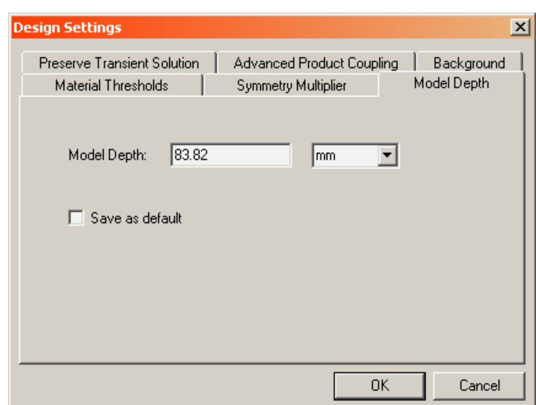
- In the Mechanical tab:
  - enter *3000 rpm* for the speed.
- Click OK to validate the setting of the Band object.

- Right mouse click on **Model** in the Project tree, then select the menu item *Set Symmetry Multiplier*



- Since we model 1/8th of the motor (our model spans on 45°), Enter *8*. The force, torque will be rescaled to take into account the full model.

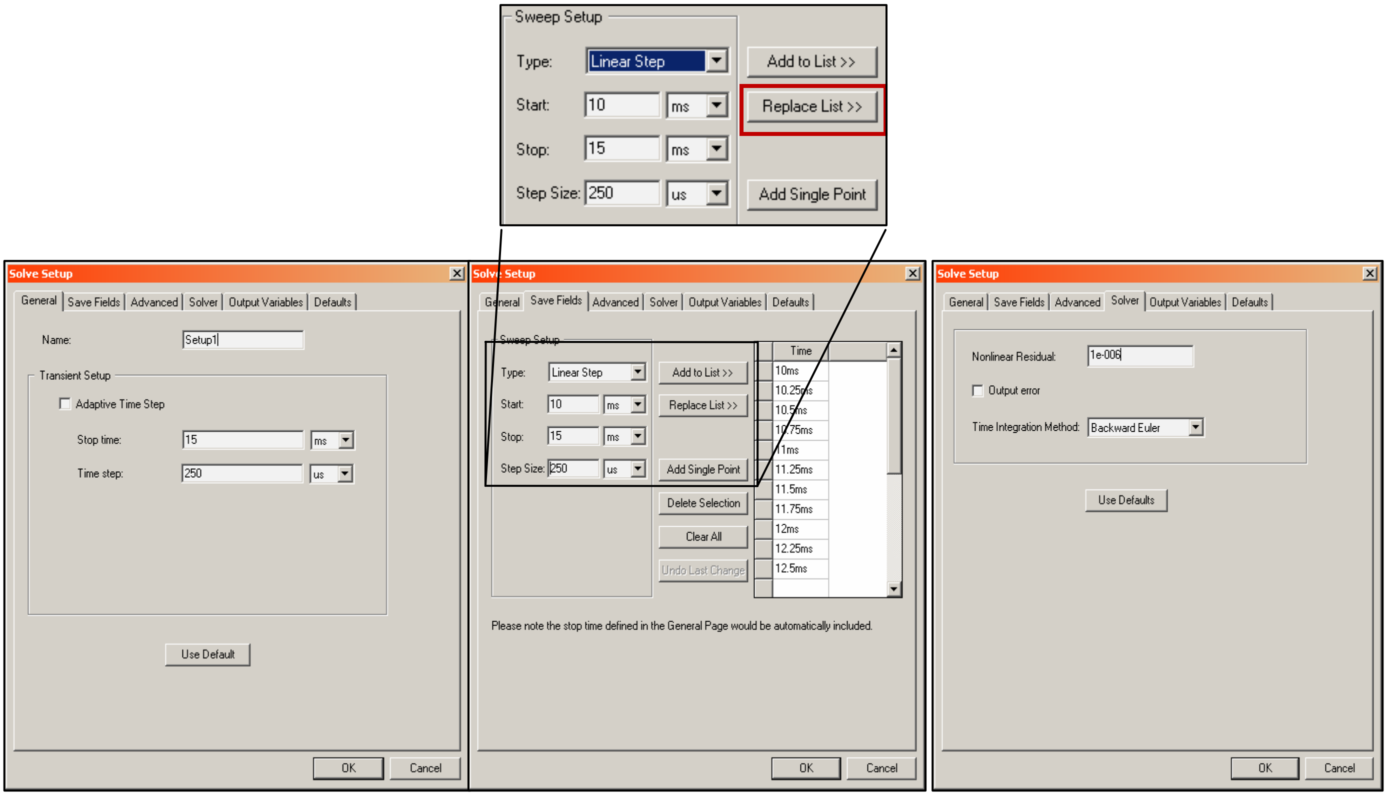




- Select the *Model Depth* tab. Enter *83.82mm* for the motor depth. All quantities will be automatically be rescaled to the correct size.
- Accept the setting

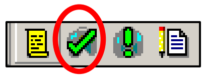
Add an Analysis Setup

- Right mouse click on Analysis in the Project tree and select Add Solution Setup:
  1. On the *General tab* enter the stop time and the time step. At 3000 rpm, a revolution takes 20ms (3000 rpm means 50 revolutions per second or 1/50 s for one revolution) . To achieve reasonable accuracy, we want to have a time step every 1 or 2 degrees. In this study, to have faster results, we use a time step of *250 us*; it corresponds to 4.5 degrees.
  2. The total simulation time is set to *15ms*
  3. On the *Save Fields* tab
     - Select Linear Step
     - For Start, put 10ms
     - For Stop, put 15ms
     - For Step Time, put 250us
     - Click on Replace List
  4. In the *Solver Tab*, set the Non linear residual to *1e-6.*

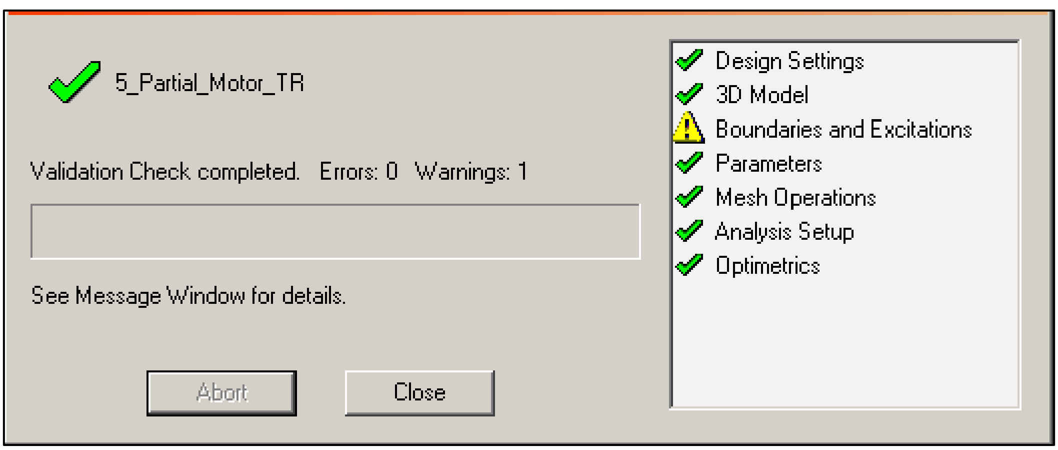


Solve the problem

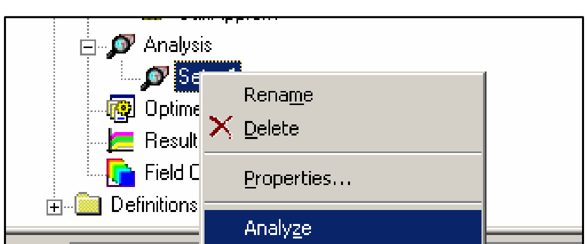
- The setup is completed. Check the project using the Validate button



- Maxwell checks the geometry, excitation definitions, mesh operations and so one. The model is validated but some Warning is displayed in the message box:
  - Eddy effect are not taken into account in our design which is what we decided

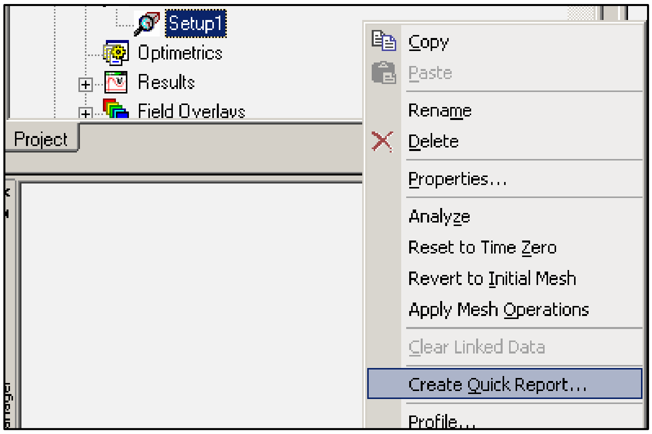


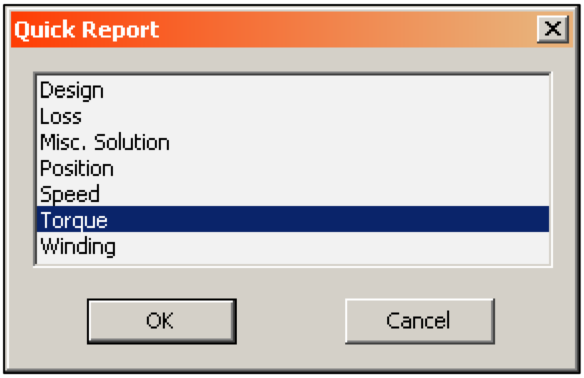
- Select the Analysis *Setup1* in the project tree, right mouse click and select *Analyse*



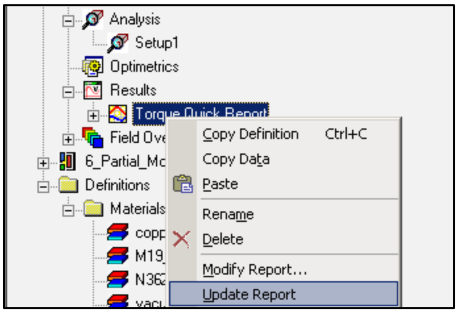
Post Processing

- The full simulation takes some minutes.
- The mesh size appears in the profile of simulation. To display the profile, select the Analysis Setup, right mouse click and select *Mesh Statistics*. The mesh statistics are available in the corresponding tab
- Performance curves can be displayed during the simulation.
- *Torque versus Time*. Select the menu item *Results* in the project tree, right mouse click, then select the menu item *Create Quick Report*



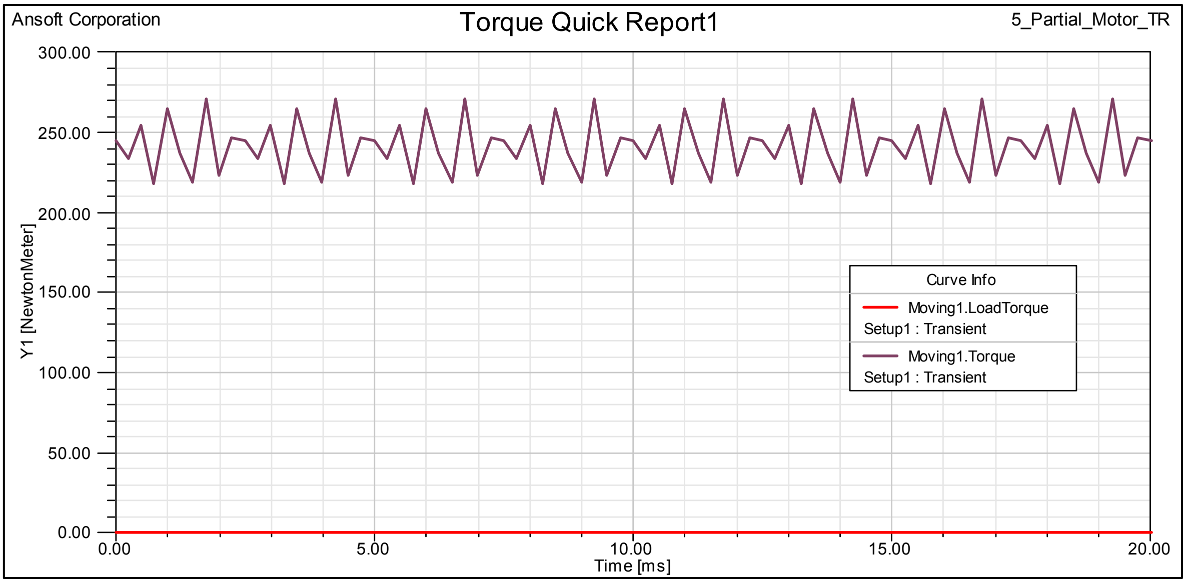


- Choose *Torque*
- The Torque up to the current time is displayed. As the simulation continues, you can update the plots: right mouse click on the *Torque Quick Report* entry and select *Update Report*



Post Processing (Cont'd)

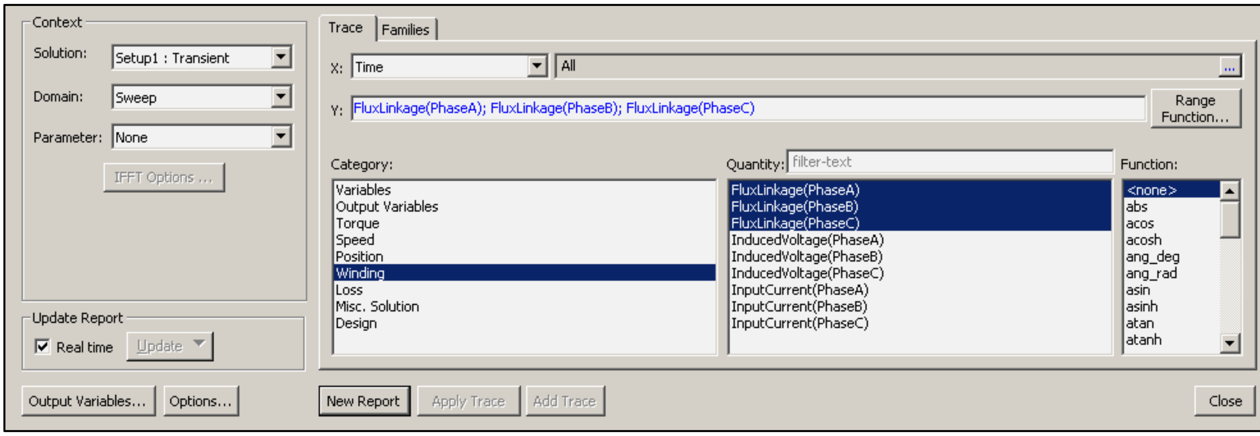
- At the end of the simulation the Torque looks like below



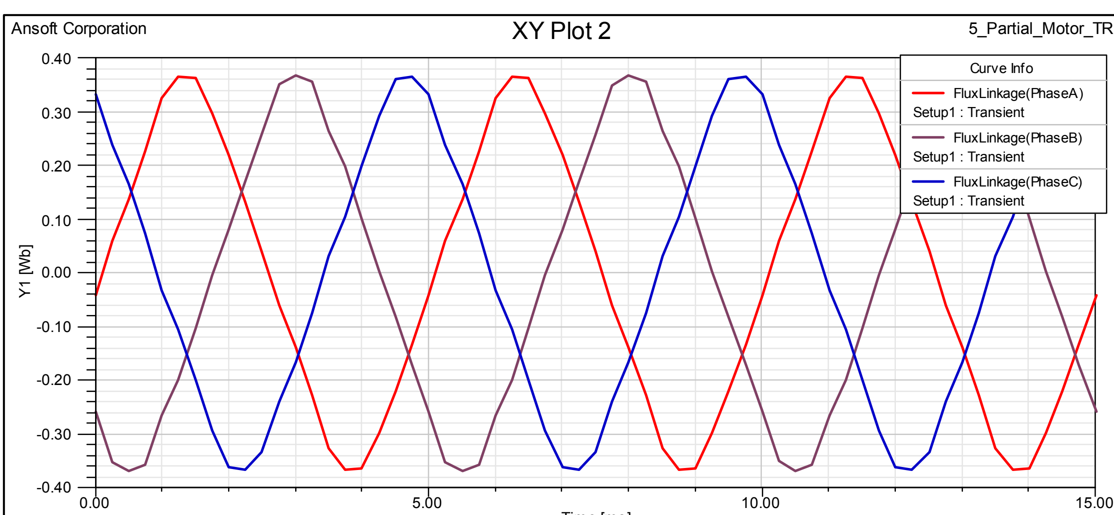
- The *LoadTorque* (in red) is zero as we are in motor mode.
- We can see that there are a lot of ripples in the Torque. The ratio between the torque and the torque ripples is almost 10 percent. This is due to the unique structure of the IPM motor (Internal Permanent Magnets). To limit the ripple, some manufacturers modify slightly the rotor shape around the magnets or add a second layer of internal magnets. Also the control strategy plays a big role into preventing the ripples.
- The torque value is around 240 N.m. This value is compatible with measurement.
- The peak torque for this motor is about 400 N.m

- *Flux linkage versus Time*. Select the menu item *Results* in the project tree, right mouse click, then select the menu item *Create TransientReport > Rectangular Plot*.

Post Processing (Cont'd)

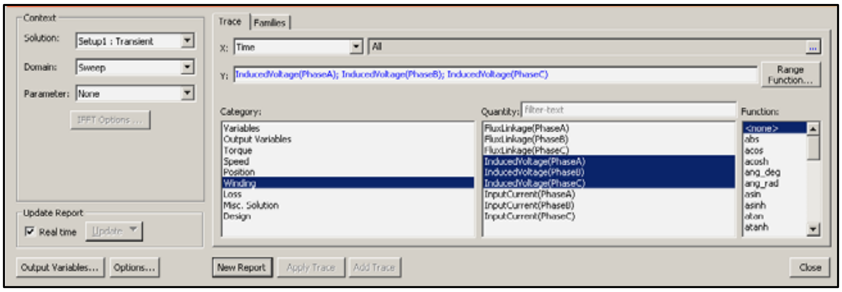


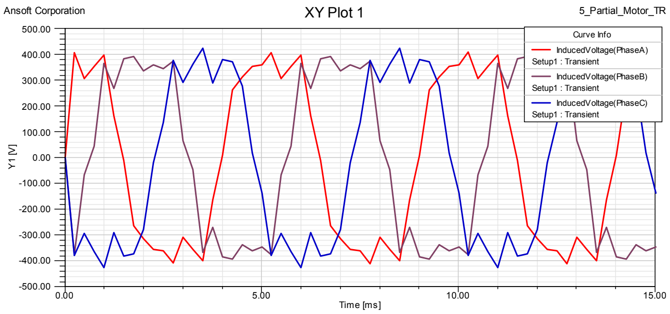
- To include the flux linkage for each coil:
  1. Select Winding in the *Category* Column
  2. Select *FluxLinkage(PhaseA), FluxLinkage(PhaseB), FluxLinkage(PhaseC)* in the *Quantity* column
  3. Select *New Report*
  4. Select Close



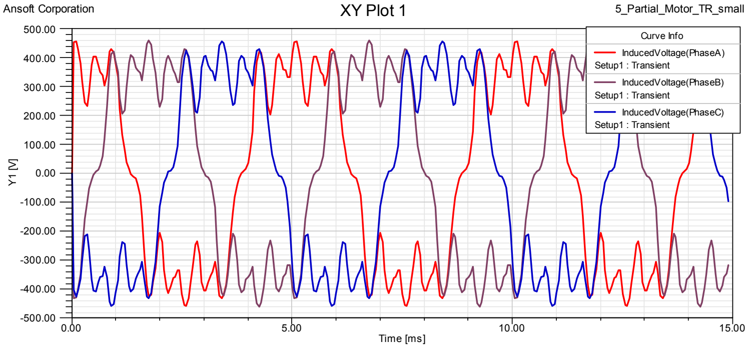
Post Processing (Cont'd)

- *Induce Voltage versus Time*. Select the menu item *Results* in the project tree, right mouse click, then select the menu item *Create TransientReport* > *Rectangular Plot*.
- Use the same method to plot the Induced Voltage



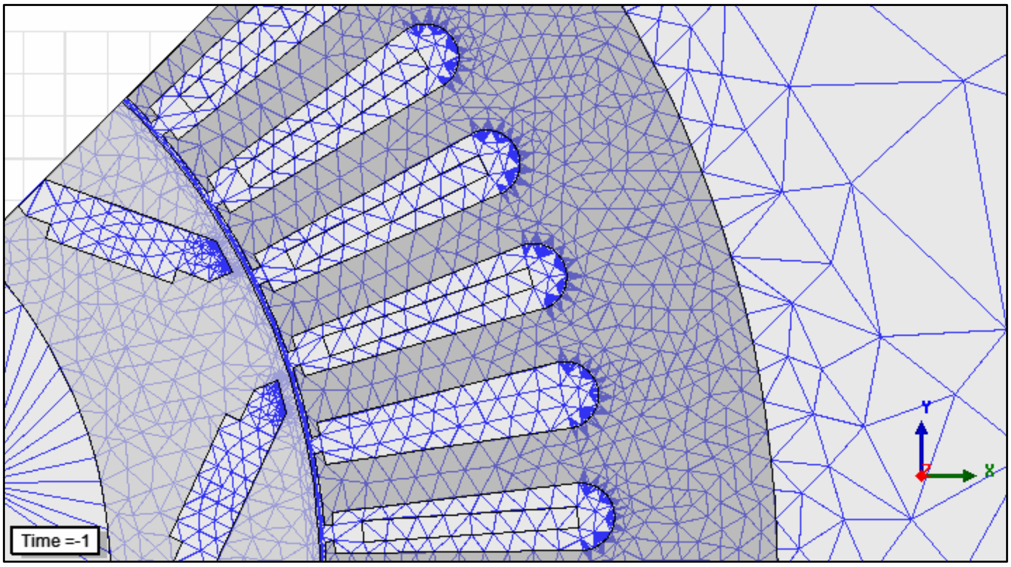


- The curves are not really smooth. The reason is that the time step is too high. As the induced voltage is a derived quantity, Maxwell needs to derive the total flux ; the time steps is way to high to have accurate Induced Voltage. If you re run the simulation with a time steps of 50us (instead of 250 us), the Induced Voltage will have a more realistic shape:



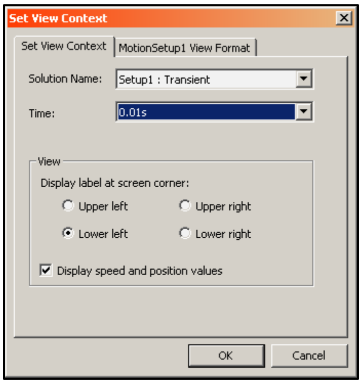
Post Processing (Cont'd)

- **Plot the Mesh.** Select all the object, right mouse click and use *the Plot Mesh* button



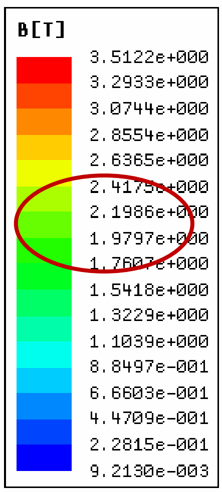
- **Plot magnetic flux density.**
  1. Select the menu item View > Set Solution Context or double click on the "Time=-1" icon in the modeler window

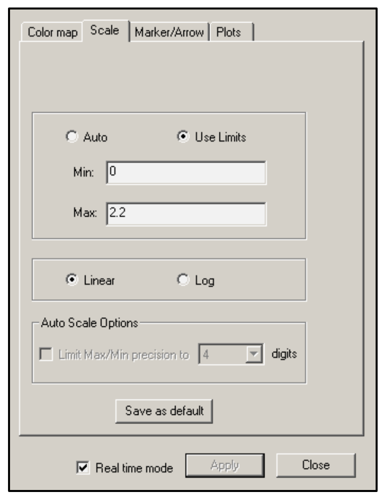
  2. Select the time **0.01s** from the pull down menu



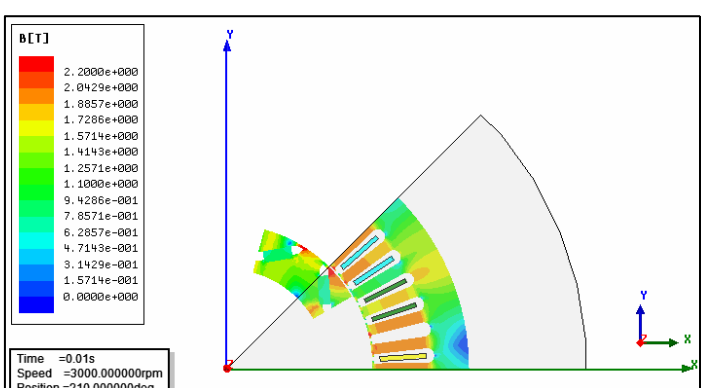
Post Processing (Cont'd)

3. Select the *Stator, Stator2, Stator3, PM1 and PM2* objects. Right mouse click and select *Fields > B > Mag B*
4. Accept the Setting
5. The B field at 0.01s is displayed.
6. Change the scale by double clicking on the Scale aera



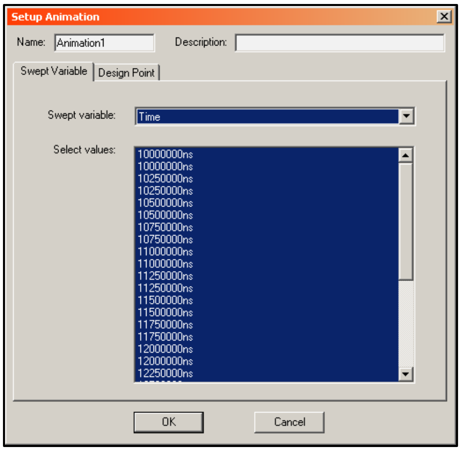


7. Go to the Scale Tab and enter *0* for min and *2.2* for max
8. Close the Window

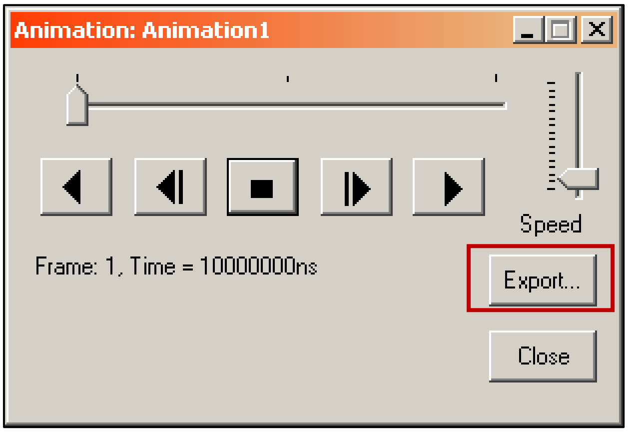


Post Processing (Cont'd)

- **Plot magnetic flux density (Animation).** It is possible to animate the fields. Select Maxwell2D > Fields > Animate.
  - Make sure that the sweep variable is Time
  - Select the time values
  - Accept the setting



- The animation is displayed once the frames are calculated.
- You can export the animation using the *Export* button from the animation button

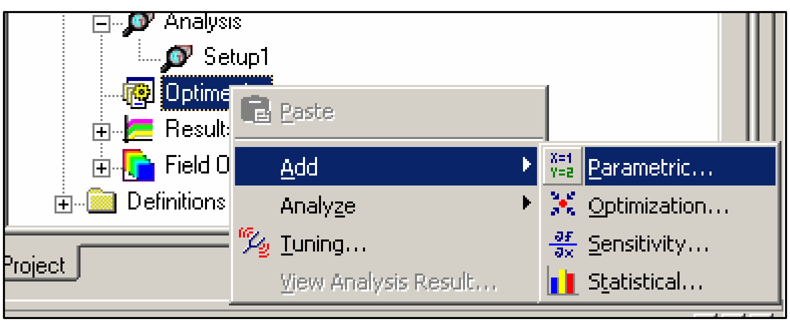


Parametric Study

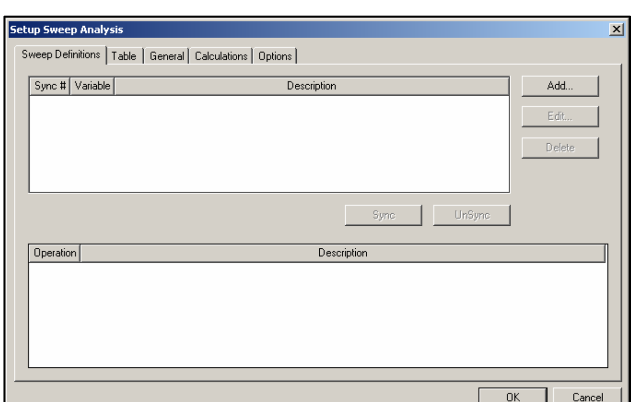
- The setup that has been solved was with a load angle of 20 deg. If the load angle is modified, the simulation has to be restarted.
- A parametric sweep of the load angle will therefore take a long time. We can propose two approaches:

  - Realize an Equivalent Circuit Extraction of the motor. This method requires the combination of parametric sweeps in magneto-static and the circuit simulator Simplorer. We will not discuss this method in this write-up.
  - Realize a parametric transient simulation. To cut the simulation time, the use of the Distributive Solve is necessary. This is the chosen method

- Click on *Optimetrics* in the Project tree. Right mouse click and select the menu item *Add > Parametric*

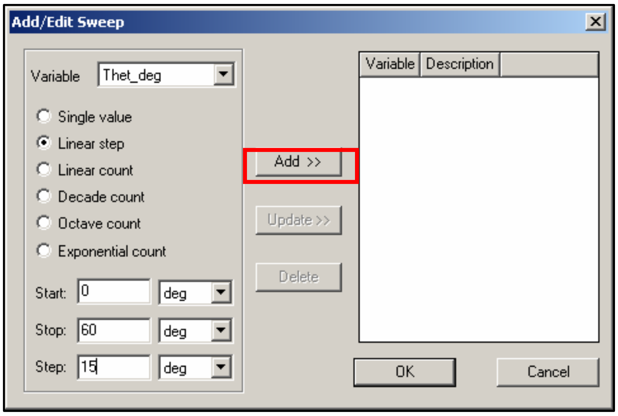


- The parametric setup panel appears



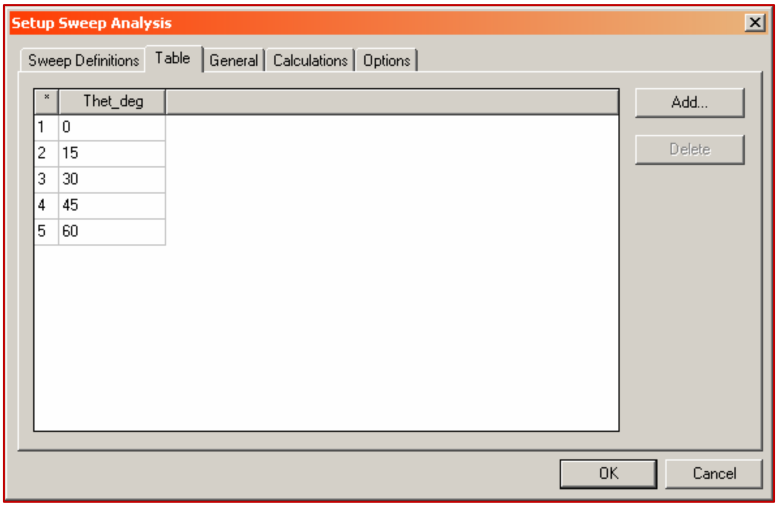
Parametric Study (Cont'd)

- Select the *Add* button to include a design variable in the sweep



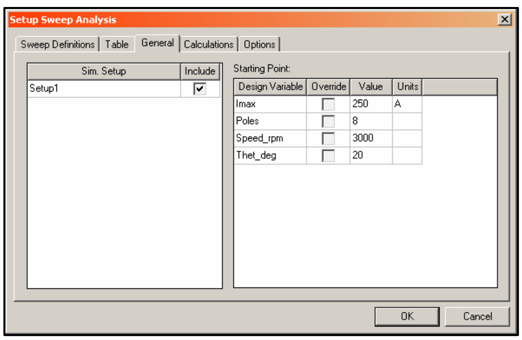
- Select *Thet_deg* from the pull-down menu:
  1. Enter *0 deg* for the first value
  2. Enter *60 deg* for the last value
  3. Enter *15 deg* for the step
  4. Push the *Add* button

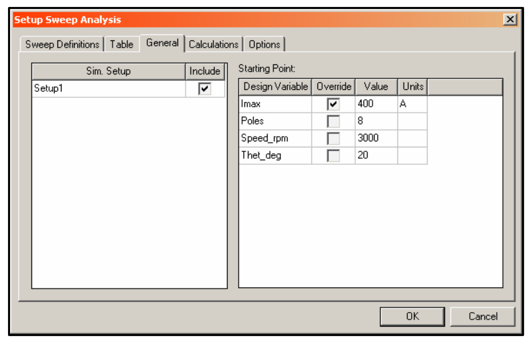
- Select the '*Table*' tab, the parametric rows are displayed



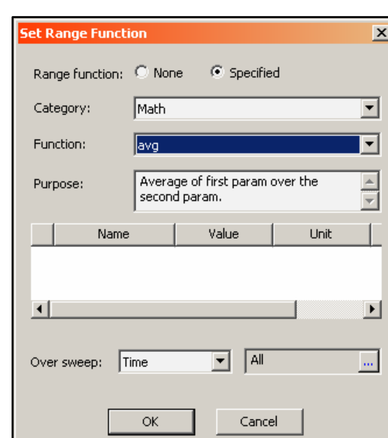
Parametric Study (Cont'd)

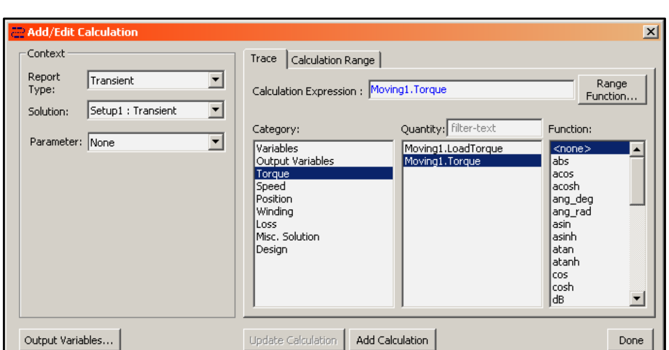
- Select the *General* Tab. This panel enables the user to change a design variable. For instance, if you wish to run the parametric sweep with a peak winding current of 400 A, select the *Override* button, and change the current value.





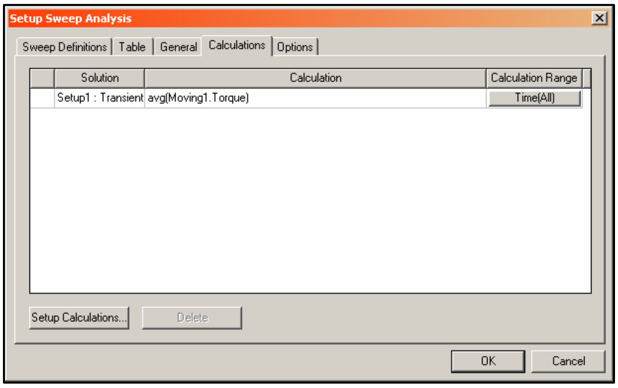
- Select the *Calculations* Tab
  1. Select the *Setup Calculations* button
  2. Under the *Category* column, select *Torque*
  3. Under the *Quantity column*, select *Moving1.Torque*
  4. select the *Range Function* button.
  5. Select the *Specified* radio button
  6. Setect the *Math* Category
  7. Select *avg* in the Function pull down menu
  8. Click on ok
  9. Click on *Add Calculation*
  10. Click on *Done*





Parametric Study (Cont'd)

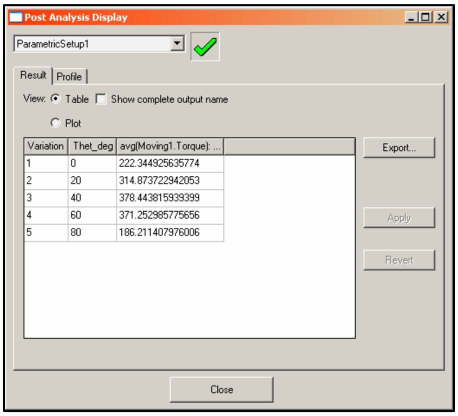
- The sweep setup panel contains the desired quantity



- In the Options tab, you can choose to save fields and meshs for all the variations
- Accept the setup

- **Run the parametric sweep.** To run the sweep, select the *Parametricsetup1*, right mouse click and select the menu item *Analyse*

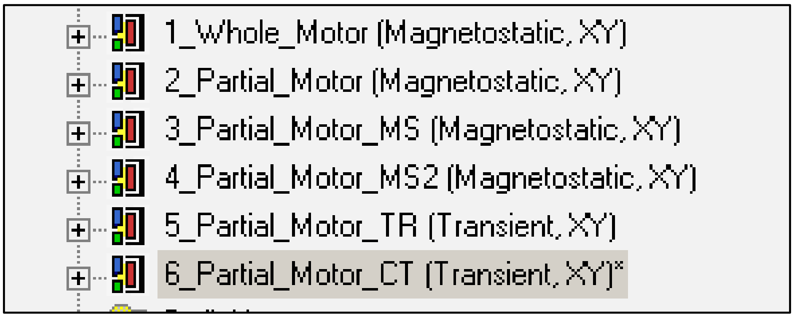
- **Results.** Right mouse click on *Parametricsetup1*, and select *View Analysis Result*



- All the plots are now available for any variation

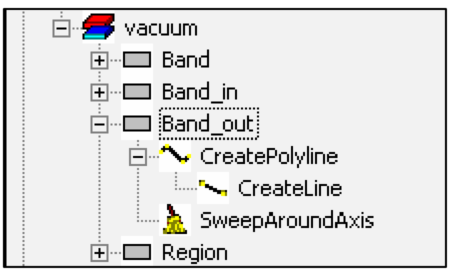
COGGING TORQUE

- The Cogging Torque corresponds to the torque due to the shape of the teeth and the permanent magnets, when all the coils excitations are 0. The torque is a very small value in regard to the full load torque. Its computation is very sensitive to the mesh, as its value is in the same order of magnitude of the mesh noise.
- To compute accurately the cogging Torque, one could solve a parametric sweep in Magnetostatic (the input parameter being the angle between rotor and stator). This method will not lead to excellent results as the error due to the mesh will be different for each position (the mesh will change for every row).
- The preferred method is the use of the transient solver with motion:
  - We will move the rotor at the speed of 1 deg/s
  - The mesh will remain unchanged for all the positions thanks to the Band object : the mesh inside the Band object will rotate with the rotor
  - Each time step will be independent of the other
  - The adaptive mesh will not be used therefore the simulation time will be shorter
- Save the project. Click on the Maxwell design '*5_Partial_motor_TR'* , right mouse click and select 'Copy'.
- Click on the project name, right mouse click and select Paste. Change the copied design to '*6_Partial_motor_CT*



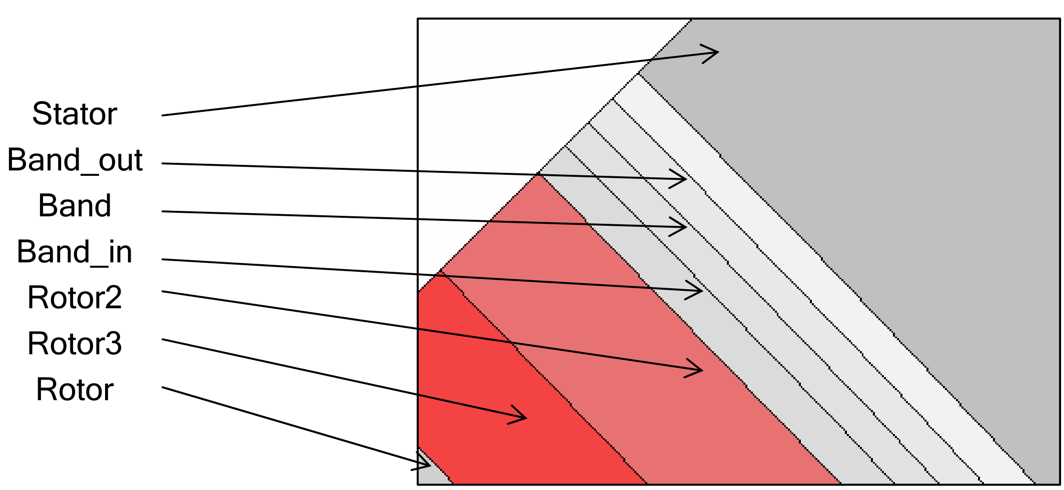
Creation of an air Object

- We derive the set up for the cogging torque calculation from the Full load setup. We will change the speed, the excitations and some meshing operations.
- Since the mesh has to be well defined in the air gap, we will add an object so that we have enough layers of element:
  1. Copy and Paste the Object Band_in
  2. Rename the object Band_in1 into Band_out.



  3. Expand the history tree of Band_out and change the CreateLine command:
     - Replace 80.4 ,0 ,0 by 80.75 ,0 ,0 mm

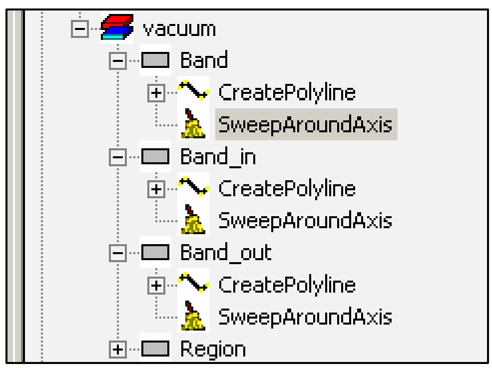
- This create a third layer in the air gap



Increase the segmentation of air objects

- For the dynamic analysis, the ojects *Band*, *Band_in* had one segment every degre. In order to reduce mesh error, we reduce the span of each segment.

- Expand the history tree of the objects *Band*, *Band_in* and *Band_out*.
    1. Double click on the *SweepAroundAxis* command of the *Band* object

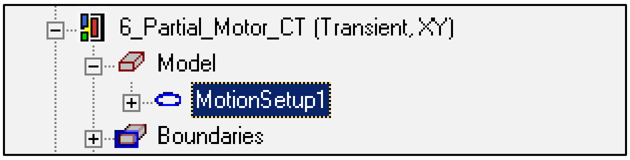


    2. Change the number of segments from *45* to *135*
    3. Repeat 1-2 for the objects *Band_in, Band_out*

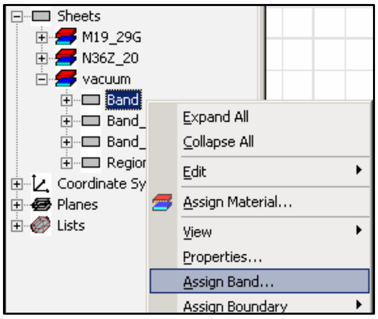
Name	Value	Unit	Evaluated Value
Command	SweepAroundAxis		
Coordinate System	Global		
Axis	Z		
Angle	45	deg	45deg
Draft angle	0	deg	0deg
Draft type	Round		
Number of Segments	135		135

Meshing Operations

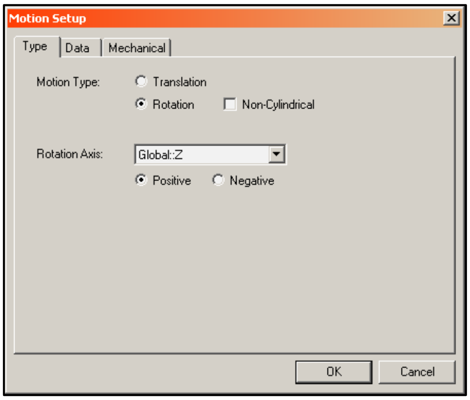
- We need to reassign the *Band*. Expand the project tree of the current design, and delete the *MotionSetup1*

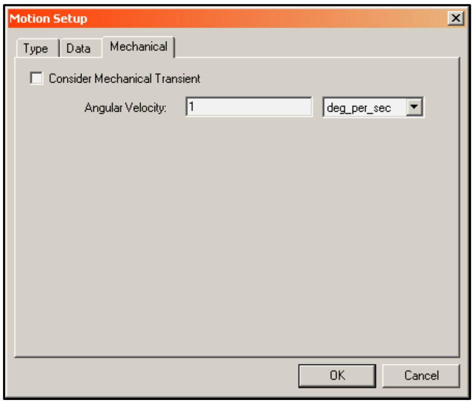


- Select the Object *Band*, right mouse click, and select *Assign Band*



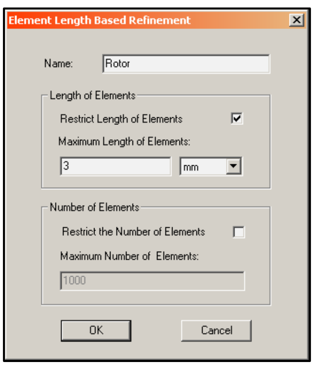
- Enter the following parameters for the Motion Setup
  - In the Type Tab, for Motion Type: *Rotation around Z axis*
  - Leave the *Data* tab unchanged
  - In the *Mechanical tab*, enter *1 deg_per_sec*
  - Accept the Setting

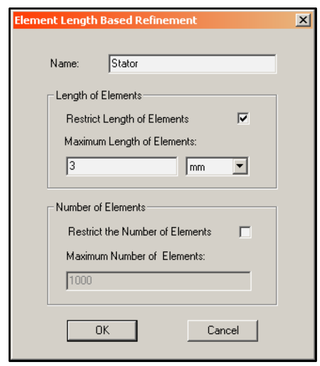




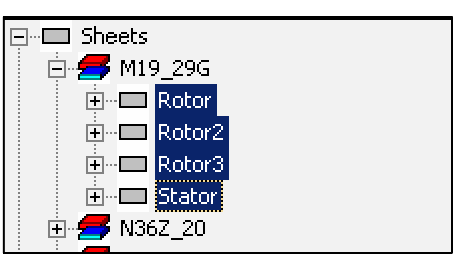
Meshing Operations

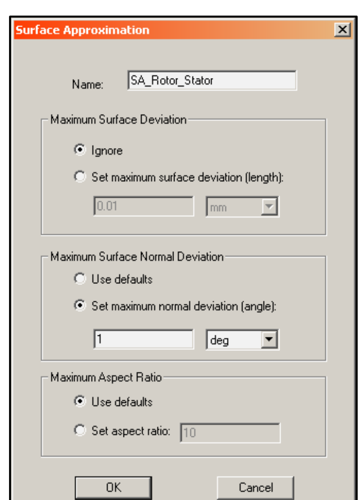
- We also need to change the meshing operations. The mesh density that was good enough to compute the full load torque won't be accurate enough for the cogging torque
- Expand the project tree, and under Mesh operations, edit the Meshing operations *Rotor*, *Stator*.
  - Change the maximum length from *4mm* to *3mm*





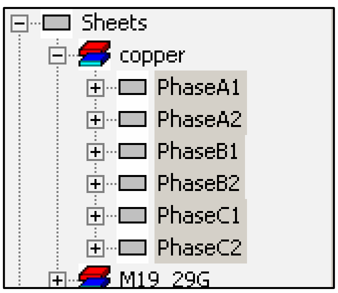
- Select the objects *Rotor*, *Rotor2*, *Rotor3* and *Stator*. Right mouse click and select Assign Mesh Operation > Surface Approximation
  - Name the meshing operation SA_Rotor_Stator
  - Set the minimum normal deviation to 1 deg
  - Ignore the other settings

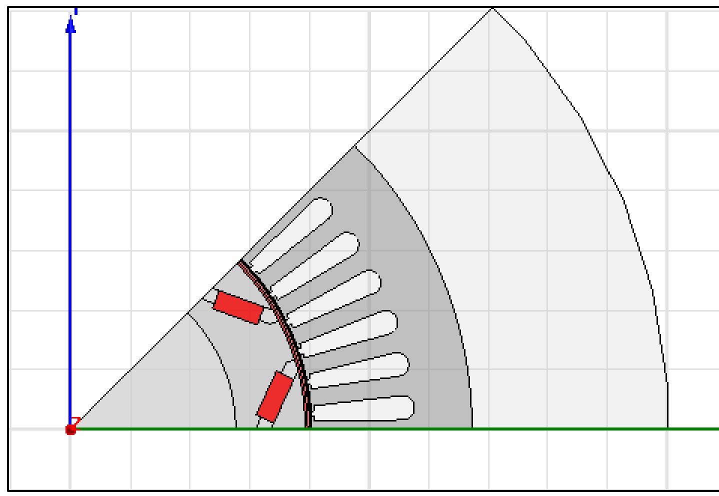




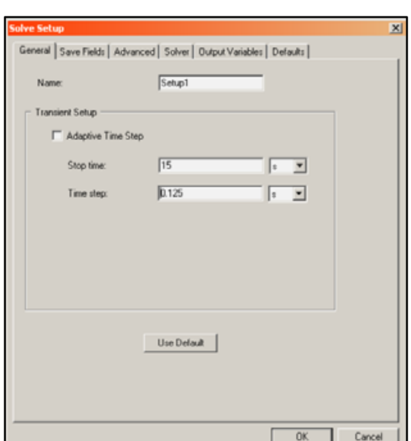
Create the Analysis Setup

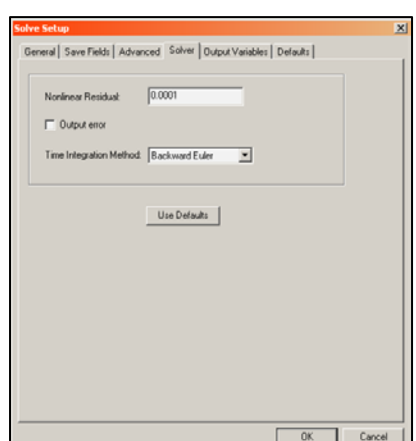
- We can delete the coils as they are not needed in the cogging torque simulation
- Select the 6 coils and delete them





- Expand the analysis setup already defined and edit it:
  - One pole pair takes 7.5 mech degrees. We will solve over ***15 s*** in order to have two periods (remember: the speed is 1 deg/s)
  - We set the time step to ***0.125 s*** to have a very smooth curve. An higher time step is still valid if you want a faster result.
  - Lower the non linear residual to 1^{e}-4.
  - Accept the Setting

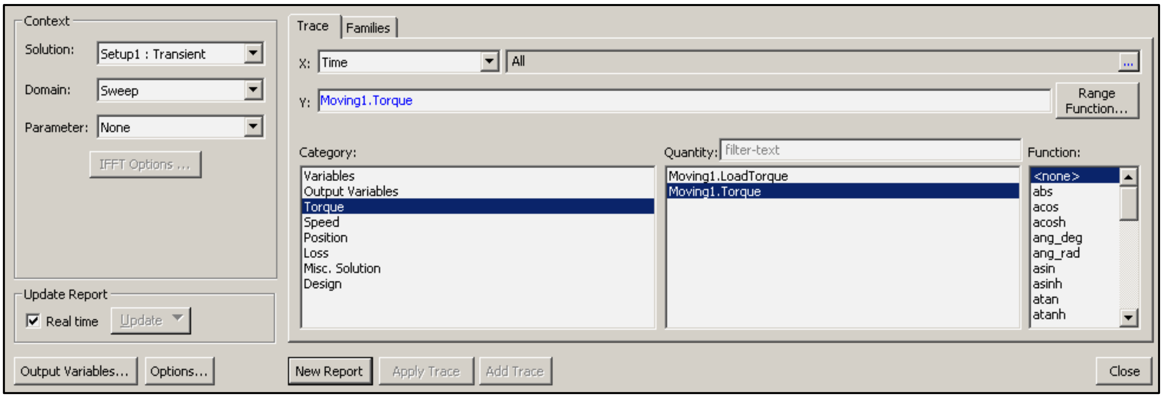




Analyse

- From the project tree, right mouse click on *Setup1*, and select *Analyse*.
- It takes a couple of minutes to solve

- From the project tree, right mouse click on *Results*, and select *Create Transient Report > Rectangular plot.* The Trace window pops up
  - From the *Category* column select *Torque*
  - From the *Quantity* column, select *Moving1.Torque*
  - Select *New Report*
  - Select *Close*



- The Torque trace appears. As expected, the cogging torque is periodical. The peak value is about *1.75 N.m*.