SIEMENS
Ingenuity for life

Simcenter 3D Low Frequency EM Tutorial #1

Interior Permanent Magnet Traction Motor

Version: Simcenter 3D 2020.1

SIMCENTER

Table of Contents

Table of Figures

1. Introduction

This tutorial introduces the 2D workflow of a PM electric traction motor in the Simcenter 3D Low-Frequency EM environment. It will cover the pre-processing setup of the motor and the post-processing analysis of the typical results (i.e. the back EMF and torque at the rated operation point).

It is important to note that in low-frequency electromagnetic analysis, the device under analysis needs to be enclosed with a fluid material (in most cases, air) that defines the computation domain. Applying the proper boundary conditions to the device results in a unique solution of the problem.

Specific to motion problems, and in particular, electric machines, the airgap is the critical region between the stationary stator and the rotating or moving rotor. The airgap needs to be divided into 4 layers for the accurate computation of the electromagnetic forces, hence torque.

The premise is that the motor under analysis is part of a complex CAD model in the Simcenter 3D CAE environment. However, for radial-flux motors, 2D analysis is sufficient. Therefore, the model should be reduced from a complex 3D system model to a 2D model as illustrated in the following figure.

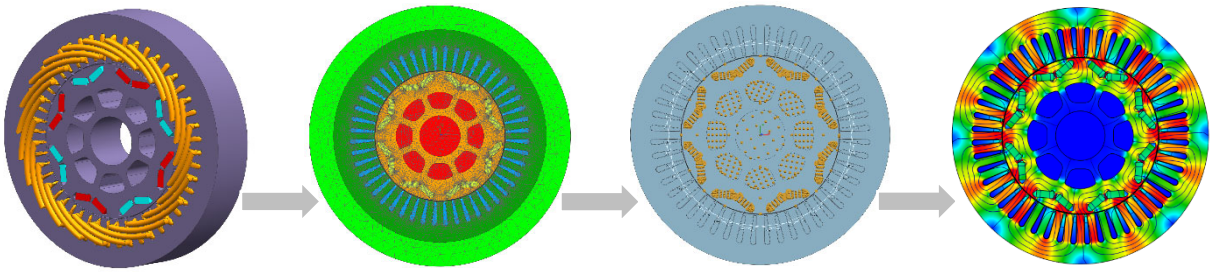

Figure 1: IPM tutorial model in Simcenter 3D 2019.1

Estimated duration: 1.5 hours

2. CAD Model Pre-processing

In this section, the CAD model is prepared for finite element discretization and EM simulation setup. This involves adding the airgap regions and the enclosing air region. It is recommended that you always create the airgap region drawings first, before using the automatic method of making the enclosing air region.

In the Start model folder, there is a part file "*EV Traction motor_2D_Tutorial.prt*" that includes the 2D drawing of the motor. We will start from this model.

- Launch Simcenter 3D by double-clicking the desktop icon
- Click File|Open -- in Open window, navigate to the *Start model* folder, select Part File(*.prt) as Files of type and select "*EV Traction motor_2D_Tutorial.prt*". Click OK.
- The "*EV Traction motor_2D_Tutorial.prt*" includes the 2D drawing of the traction motor as shown below:

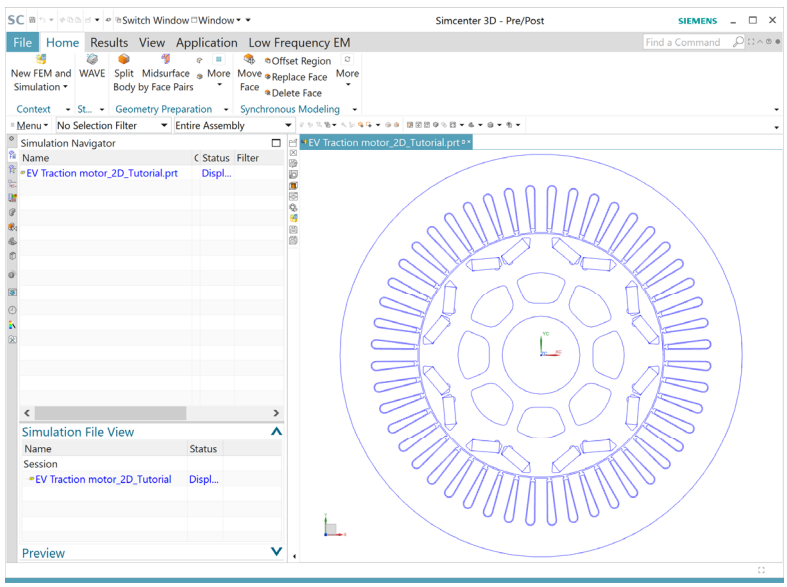

Figure 2: IPM 2D Drawing part file

2.1. Creating the airgap layers

The airgap region of a motor is where the stator and rotor fields interact to produce torque. Therefore, it needs to be adequately refined for accurate computation of the fields. Four layers are recommended for the accurate computation of the forces and torques.

- Right-click *EV Tranction motor_2D_Tutorial* in Simulation Navigator and select New FEM and Simulation

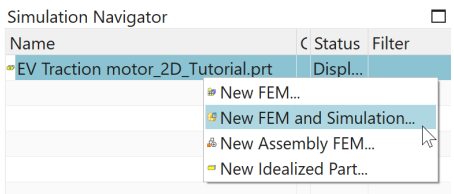

Figure 3: Create new FEM and Simulation

- In the New FEM and Simulation setup window:
  - Check the option Create Idealized Part
  - Set "Polygon Body Resolution" to High
  - Set Solver to Simcenter MAGNET
  - Set Analysis Type to Electromagnetic-2D Translational

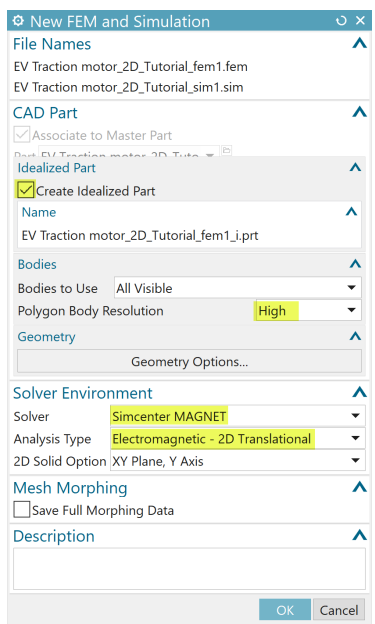

Figure 4: Setup for new FEM and Simulation

- After clicking OK, the solution setup window will pop up. We will return to the solution setup later so, for now, click Cancel.
- Double-click "*EV Traction motor_2D_Tutorial_fem1_i.prt*" in the Simulation Navigator to make it the work part.

- Click Menu|Insert|Sketch – from Create Sketch dialog, click OK and then draw three circles centered at the origin with radii of 80.3875 mm, 80.575 mm and 80.7625 mm.

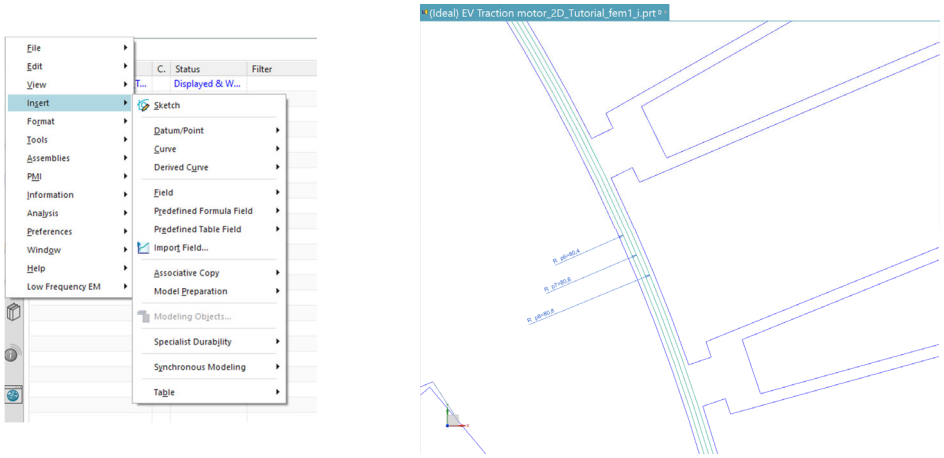

Figure 5: Insert Sketch in Task Environment and draw 3 circles

- Ensure that Selection Scope is set to **Entire Assembly**.

- To close the slots, zoom into one of the slots and draw a line to close it by snapping to the vertices of the existing sketch. Select the line and make 48 copies using the Circular layout of the Pattern Definition.

Figure 6: Close the slot openings

- Click "Finish" under the Home|Sketch commands group to exit the sketch mode.

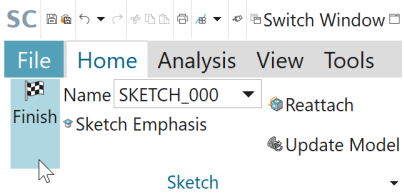

Figure 7: Finish Sketch

2.2. Create air region

The air region enclosing the model is created automatically.

- Click Pre/Post from Application|Simulation tab to exit the modeling mode.
- Click "Create Air Region" under the Low Frequency EM tab from the menu ribbon
- In the popup window, select 2D & Circle and set the scale factor to 1

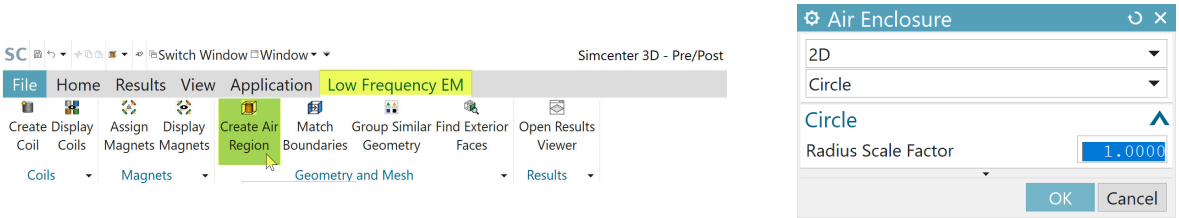

Figure 8: Create Air Region

- Click OK.

  The task to create 4 airgap regions and the enclosing air region for the 2D model is complete.

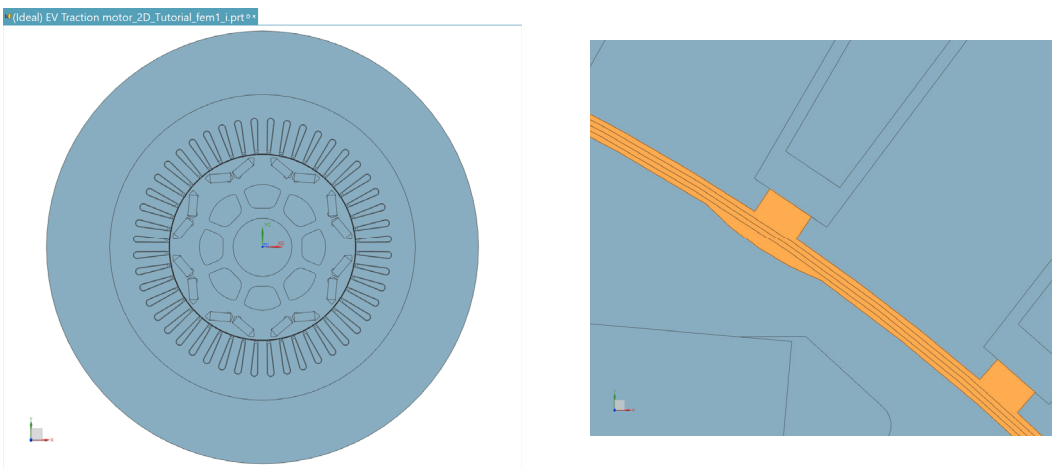

Figure 9: 2D simulation model with air regions

3. Finite Element model

In Simcenter 3D, all the operations related to mesh and materials are done in the fem file. This section will first create meshes for each parts of the motor and then assign the materials to the corresponding mesh collectors. Since this is a permanent magnet motor, the direction of the magnetization will also have to be defined.

- Right-click "*EV Traction motor_2D_Tutorial_fem1_i.prt*" in the Simulation Navigator, select Display FEM and then "*EV Traction motor_2D_Tutorial_fem1.fem*". This will make the fem the work part.
- An alternative way to make the fem the work part is to double-click the fem file name in the Simulation File View window.

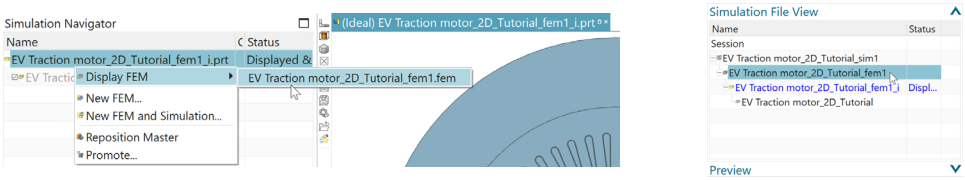

Figure 10: Make fem the work part

3.1. Prepare Materials

The materials in the model are listed below:

- Air, remesh region: Air
- Virtual air region: Virtual air
- Coil material: Copper 5.77e7
- Core material: M-19 29 Ga
- Magnet material: N42

Since different mesh collectors might have the same material, the Physical Properties table will be used to better organize the material selection.

- In Home | Properties commands group, click Physical Properties.

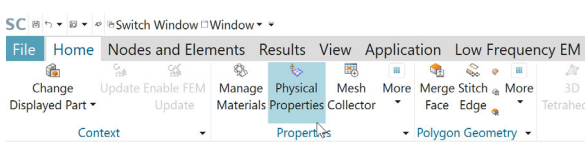

Figure 11: Physical Properties

- In the Physical Property Table Manager window, set the Type to **Solid** and Name to **Air** and click Create.
- In Solid window, click Choose material button beside the Material cell.

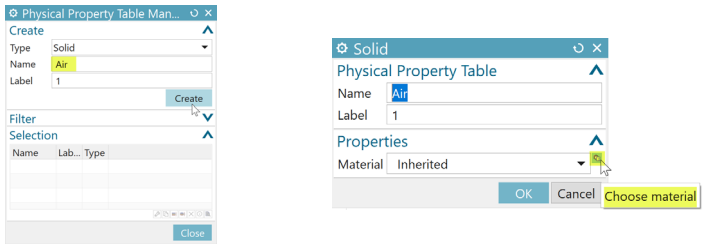

Figure 12: Create physical property table

- In the popup Material List window, select AIR and click OK.

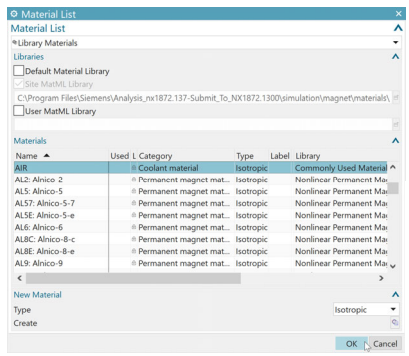

Figure 13: Material List window

- In Solid window, click OK.

The Air physical property is listed under Selection in Physical Property Table Manager window. Follow the same steps to create the other physical properties that are required:

- Virtual air region: Virtual air
- Coil material: Copper 5.77e7
- Core material: M-19 29 Ga
- Permanent Magnet material: N42

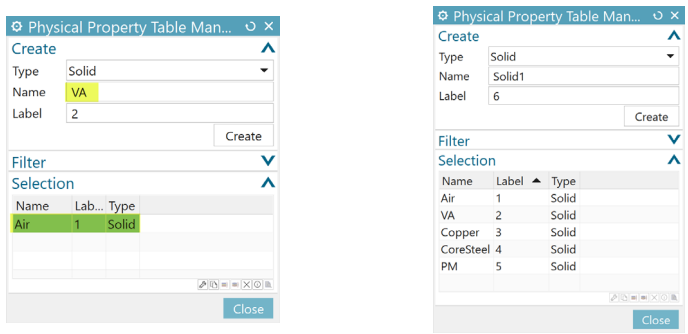

Figure 14: Physical Property table

- Once complete, click Close to finish the physical property table segment of this tutorial.

3.2. Generate Mesh

For EM analysis, a conformal mesh is required. In a 2D case, the air region tool internally uses the bounded plane with divide face method, so that it only creates one sheet body with multiple faces that represent the different parts of the model. This guarantees that the mesh will be conformal without the need for any additional operation, such as Stitching Edges.

Click Home|Mesh|2D Mesh to open 2D mesh window

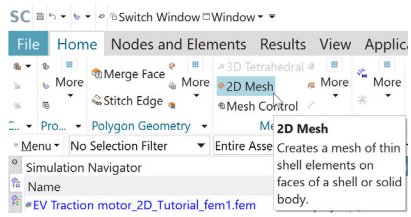

Figure 15: Physical Properties

In the 2D Mesh window, the following selections (highlighted below) are required settings for EM models:

- Element Properties type: Linear Triangle-2D Translational
- Mesh Parameters: Create Mesh For Each Face
- Mesh Settings: Export Mesh to Solver
- Model Cleanup Options: Small Feature Tolerance should be always set to 0

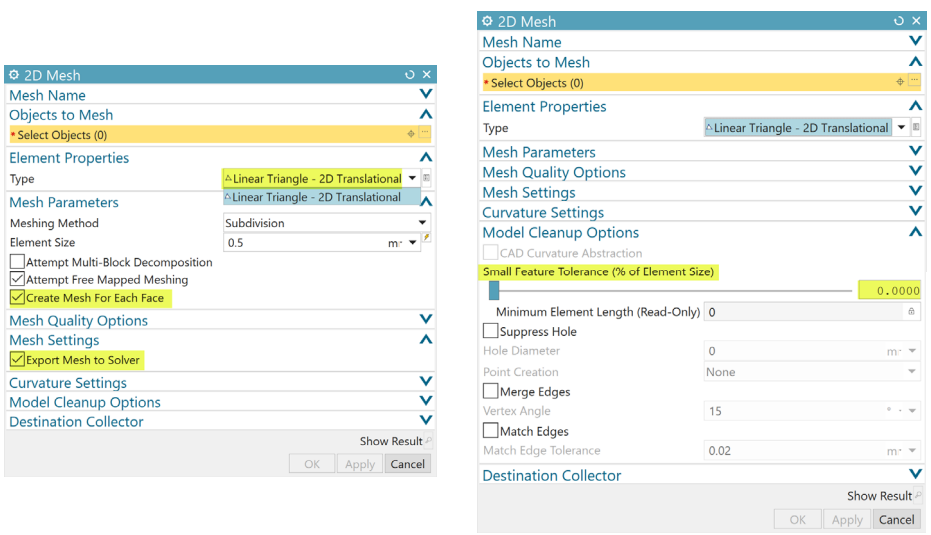

Figure 16: 2D Mesh Settings

Due to Simcenter 3D's meshing paradigm, the order of the mesh assignments is important. Where faces meet, it is recommended that you start the mesh from the face that has the smaller mesh element size. In this case, the air gap layers are critical and have the smallest regions, so they will be meshed first.

- Select two middle layer faces as Objects to Mesh. Set Element Size=0.5 mm, Surface Curvature Based Size Variation =70.

  Note: Accept all other defaults, including the required settings mentioned above. For more information about the 2D Mesh, please refer to Siemens Doc Center Simcenter help.

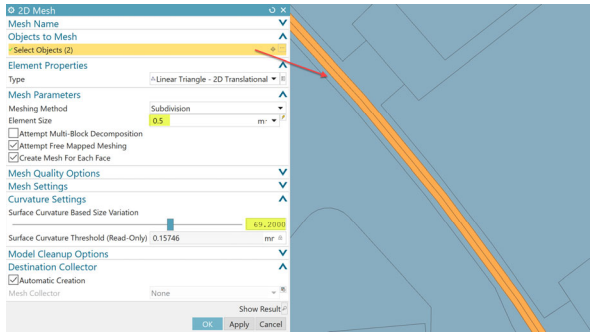

Figure 17: 2D Mesh Settings for two middle layers

- In Simcenter, the meshes are grouped by Mesh Collector. The collector can be created automatically or manually. Clicking New Collector button at the bottom beside the mesh collector cell will open the Mesh Collector window. Select Air from the drop-list of Solid Property cell and set a name for the collector (e.g. Remesh).

  Note: The Solid Property drop-down list shows all the materials that were previously created in the Physical Property table.

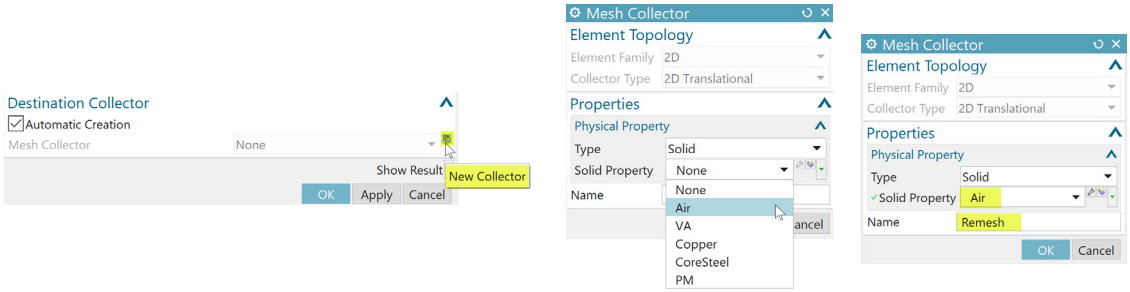

Figure 18: Mesh Collector Settings for two middle layers

- Click OK and the name Remesh will show in Mesh Collector cell.

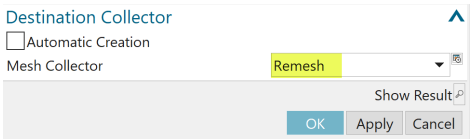

Figure 19: Mesh Collector

- Click OK to start the mesh generation process. Once complete, the 2D Mesh window closes. The meshes are displayed in the view window, whilst the Remesh mesh collector, listed under 2D Collectors in the Simulation Navigator, indicates that it consists of two meshes [i.e. 2d_mesh(1) and 2d_mesh(2)].

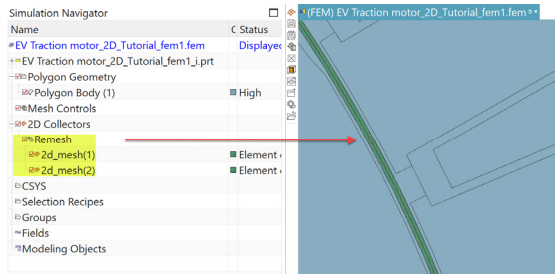

Figure 20: 2D mesh for two middle layers

- Click Home | Mesh | 2D Mesh to re-open the 2D mesh window once again.
- Select two layer faces on both side of the remesh layers as Objects to Mesh. Set Element Size=0.5 mm, Surface Curvature Based Size Variation =70. Set the mesh collector name to VA Layers and select VA as Solid Property.

Figure 21: 2D Mesh Settings for two virtual air layers

- Click OK to start the mesh generation process. Once complete, the 2D Mesh window closes. The new meshes are displayed in the view window, whilst the VA Layers mesh collector, listed under 2D Collectors in the Simulation Navigator, indicates that it consists of two meshes [i.e. 2d_mesh(3) and 2d_mesh(4)].

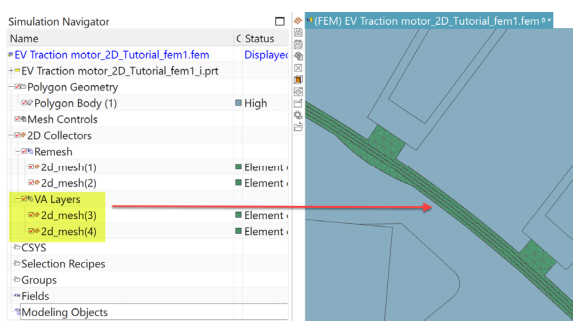

Figure 22: 2D mesh for two virtual air layers

- For a selection of multiple faces, such as the coil faces in this model (48 faces), a tool Group Similar Geometry, located in the Low Frequency EM ribbon, is very helpful.

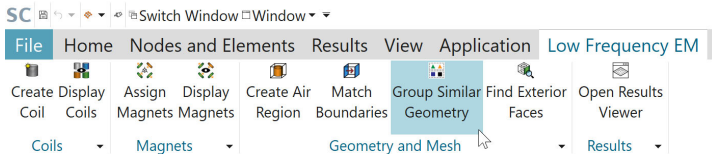

Figure 23: Group Similar Geometry tool

- Clicking Group Similar Geometry will group the similar geometry under Groups in Simulation Navigator.

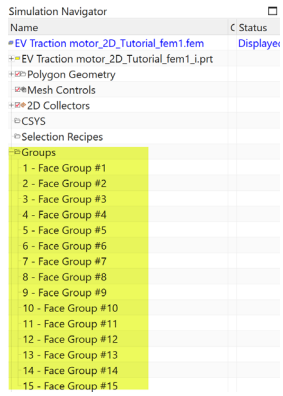

Figure 24: Similar Geometry groups

- Selecting Face Group #1 displays the corresponding faces (i.e. coil faces), highlighted in the view window.

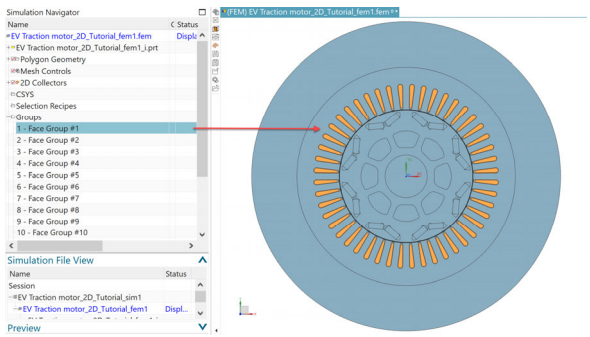

Figure 25: Coil faces group

- Click Home|Mesh|2D Mesh to re-open the 2D mesh window once again.
- Set Element Size=2.5 mm, Surface Curvature Based Size Variation =70. Set the mesh collector name to Coils and select Copper as Solid Property.

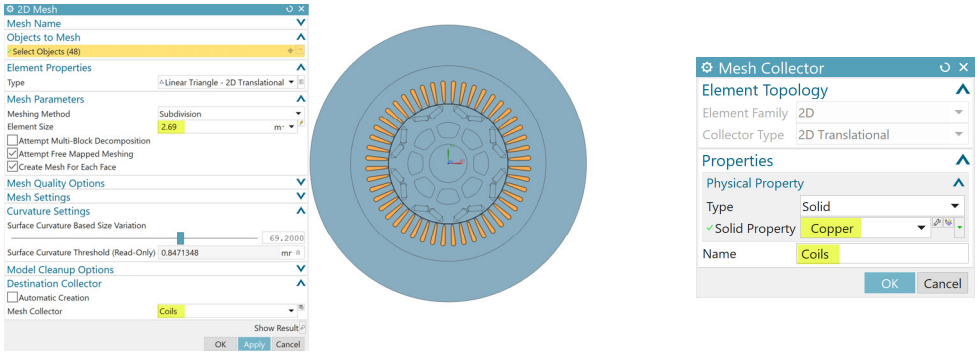

Figure 26: 2D Mesh Settings for coils

- Click OK to start the mesh generation process. Once complete, the 2D Mesh window closes. The new meshes are displayed in the view window, whilst the Coils mesh collector, listed under 2D Collectors in the Simulation Navigator, indicates that it consists of 48 meshes [i.e. 2d_mesh(5) to 2d_mesh(52)].

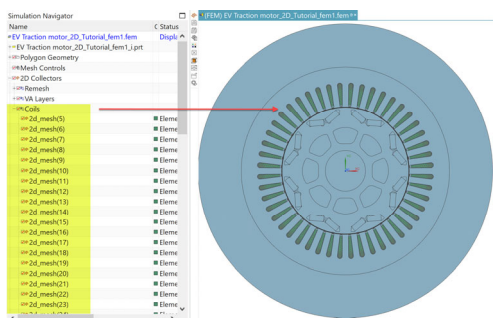

Figure 27: 2D mesh for coils

- Select Face Group #2 (the slot air regions) and click Home|Mesh|2D Mesh to open the 2D mesh window. Clicking the Automatic Element Size button generates an Element Size of 1.5 mm; accept all the other settings and create a Slot air mesh collector, selecting Air as its Solid Property.

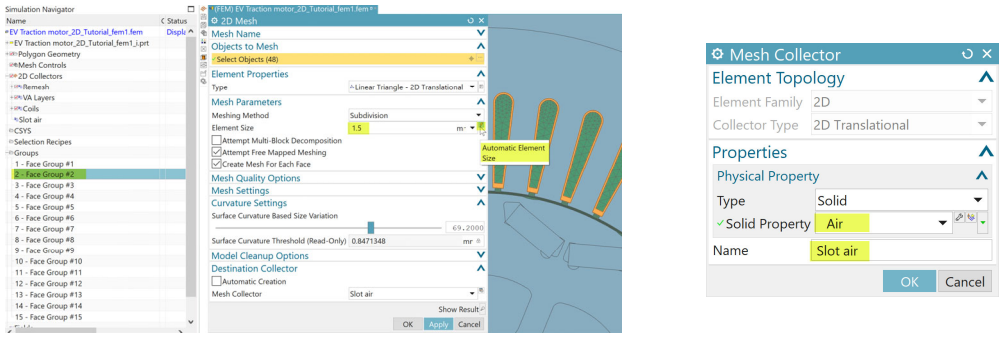

Figure 28: 2D Mesh Settings for slot air

Click OK to start the mesh generation process. After finishing, Slot air is shown under 2D Collectors in Simulation Navigator.

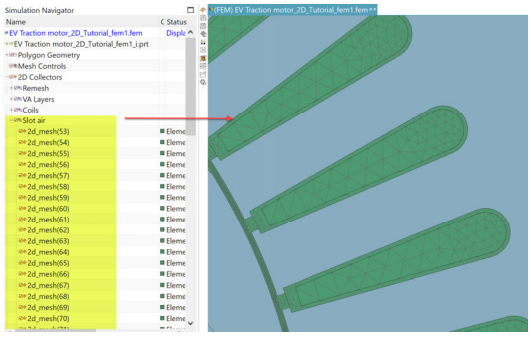

Figure 29: 2D mesh for slot air

Select Face Group #7 (the 16 permanent magnets) and click Home|Mesh|2D Mesh to open 2D mesh window. Set Element Size=1.25 mm; accept all the other settings and create a Magnets mesh collector, selecting PM as its Solid Property.

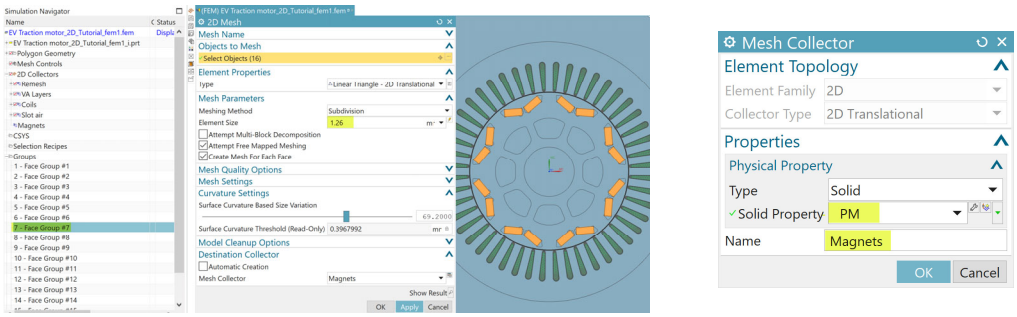

Figure 30: 2D Mesh Settings for permanent magnets

Click OK to start the mesh generation process. Once complete, the 2D Mesh window closes. The new meshes are displayed in the view window, whilst the Magnets mesh collector, listed under 2D Collectors in the Simulation Navigator, indicates that it consists of 16 meshes [i.e. 2d_mesh(101) to 2d_mesh(116)].

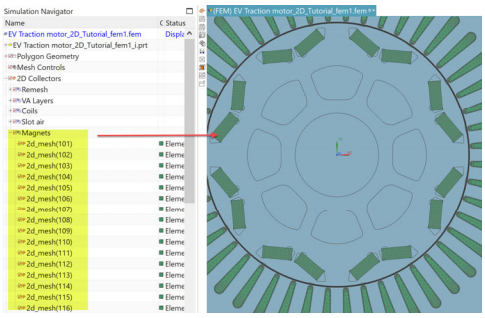

Figure 31: 2D mesh for permanent magnets

Select Face Group #6, #8 and #9 (the permanent magnet slot air regions) and click Home|Mesh|2D Mesh to open 2D mesh window. Clicking Automatic Element Size button will give Element Size=1.5 mm, keep other settings and create a mesh collector with name PM air and Air as Solid Property.

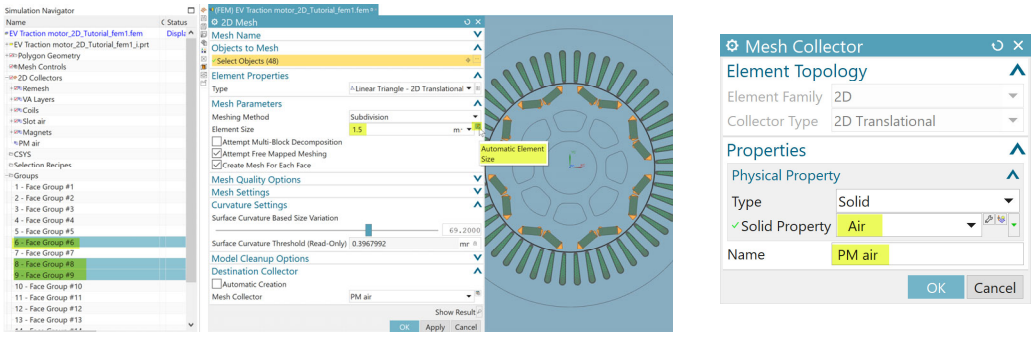

Figure 32: 2D Mesh Settings for PM air

Click OK to start the mesh generation process. After finishing, PM air is shown under 2D Collectors in Simulation Navigator.

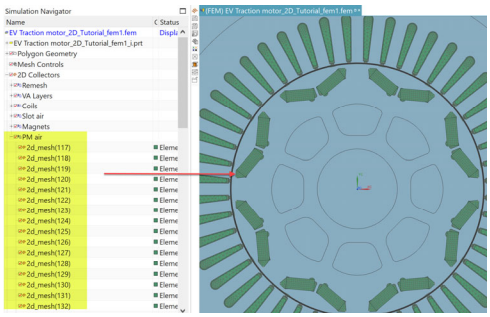

Figure 33: 2D mesh for PM air

Select stator face as Objects to Mesh. Set Element Size=6 mm. Set the mesh collector name to Stator and select CoreSteel as Solid Property.

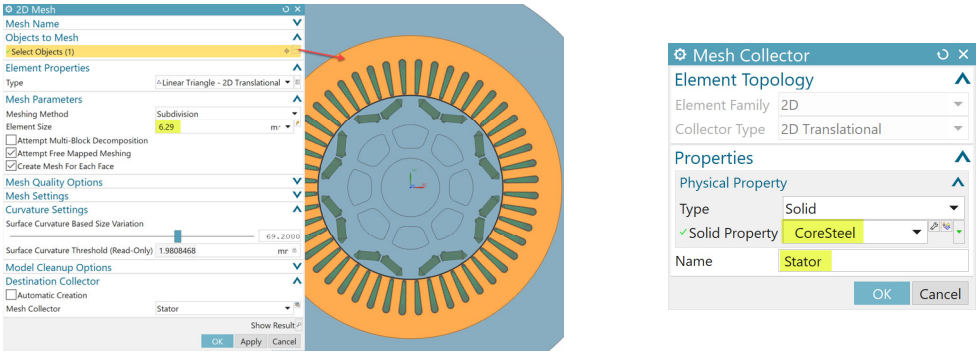

Figure 34: 2D Mesh Settings for stator

Click Apply to start the mesh generation process. After finishing, Stator is shown under 2D Collectors in Simulation Navigator.

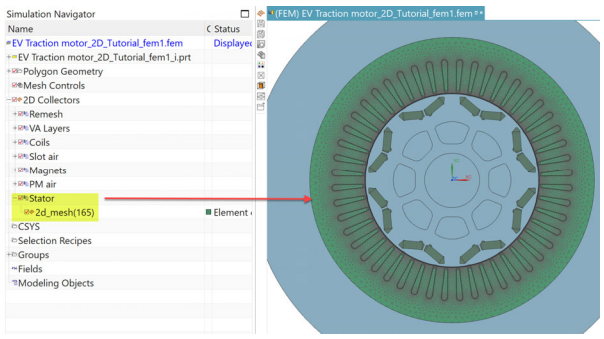

Figure 35: 2D mesh for Stator

Clicking Apply will keep 2D Mesh window open. Now, select rotor face as Objects to Mesh. Set Element Size=5 mm. Set the mesh collector name to Rotor and select CoreSteel as Solid Property.

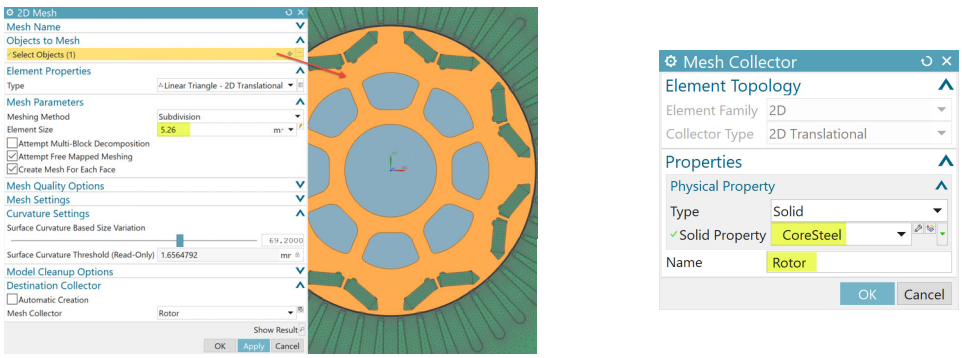

Figure 36: 2D Mesh Settings for rotor

Click Apply to start the mesh generation process. After finishing, Rotor is shown under 2D Collectors in Simulation Navigator.

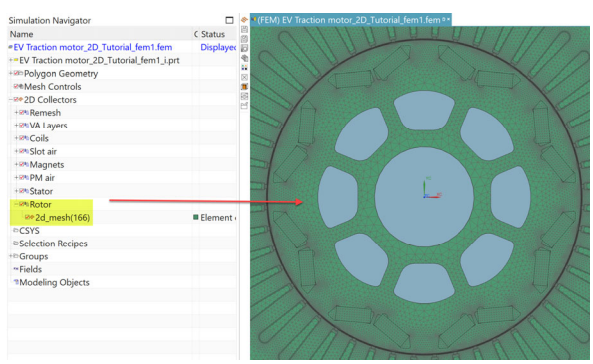

Figure 37: 2D mesh for Rotor

- Now, select the exterior face of the stator as Objects to Mesh. Set Element Size=12 mm. Set the mesh collector name to Stator air and select Air as Solid Property.

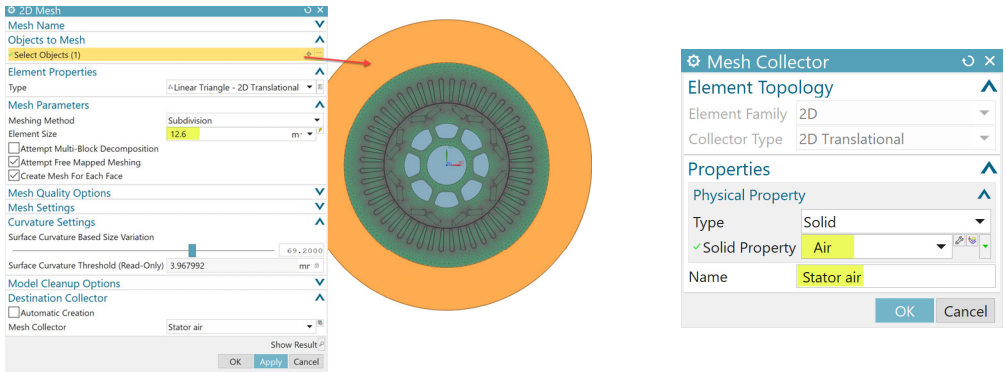

Figure 38: 2D Mesh Settings for Stator air

- Click Apply to start the mesh generation process. After finishing, Rotor is shown under 2D Collectors in Simulation Navigator.

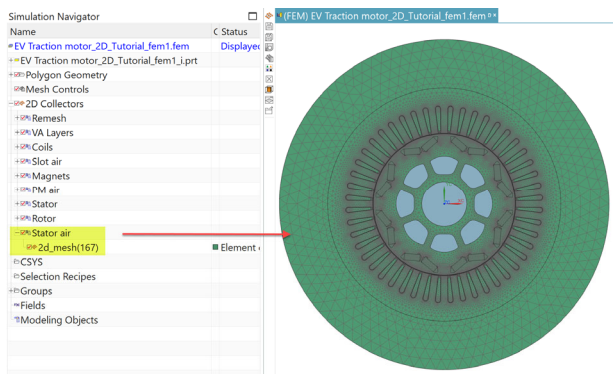

Figure 39: 2D mesh for Stator air

- Use Ctrl-A to select all the rest of the faces as Objects to Mesh. Set Element Size=6 mm. Set the mesh collector name to Rotor air and select Air as Solid Property.

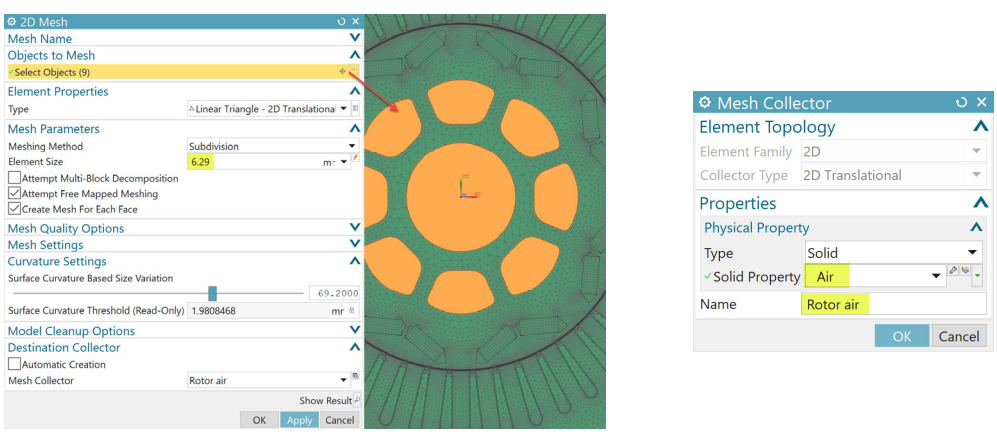

Figure 40: 2D Mesh Settings for Rotor air

Click OK to start the mesh generation process. After finishing, Rotor air is shown under 2D Collectors in Simulation Navigator.

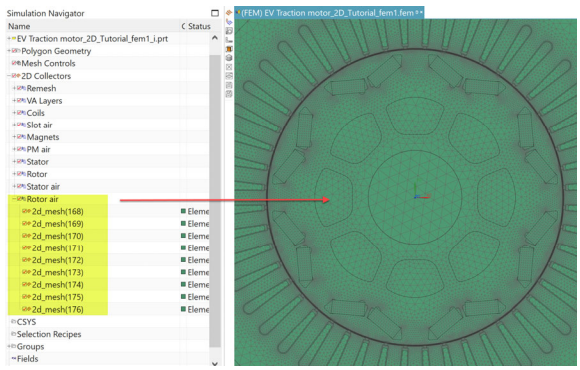

Figure 41: 2D mesh for Rotor air

To get a better view of mesh, click Home|Utilities|More and select Model Display Preferences. In Model Display window, under Element page, select Mesh Collector for Color Basis and click Set Mesh Colors button.

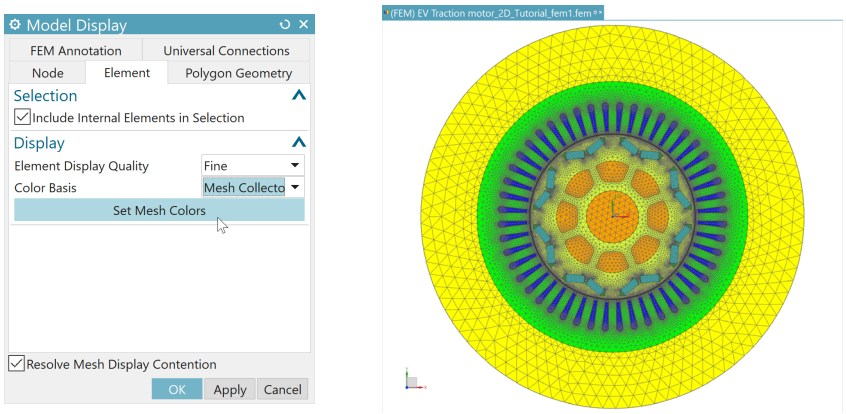

Figure 42: 2D Mesh of the model

Click File|Save|Save all to save the files.

3.3. Permanent Magnet Direction

The magnetization direction of a permanent magnet is defined in the mesh associated data. For the most common direction, uniform type, there is a tool automating the definition process using Simcenter 3D's pattern definition feature. For the motor used in this demo, the magnetization direction is uniform type.

- Uncheck all other mesh collectors, keeping only Magnets visible in the view.

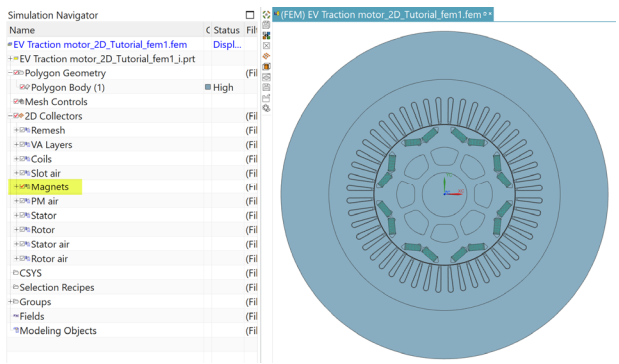

Figure 43: Show Magnets mesh only

- From the Low Frequency EM menu ribbon, click Assign Magnets.

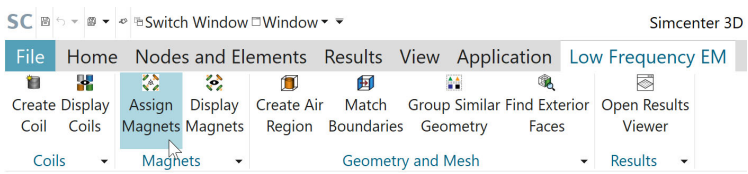

Figure 44: Assign Magnets command

- From the Assign Magnets Orientation window, select the mesh from view window as shown below.

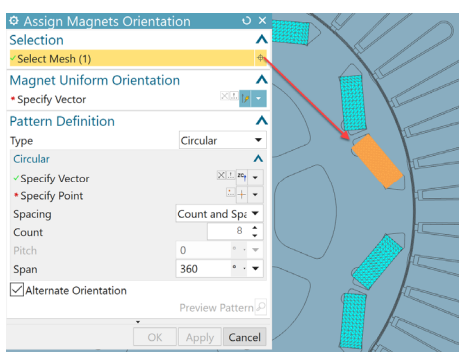

Figure 45: Select the mesh for assigning direction

- Select the Two Points method for specifying the vector, and then select the two vertices of the inner edge, as shown below:

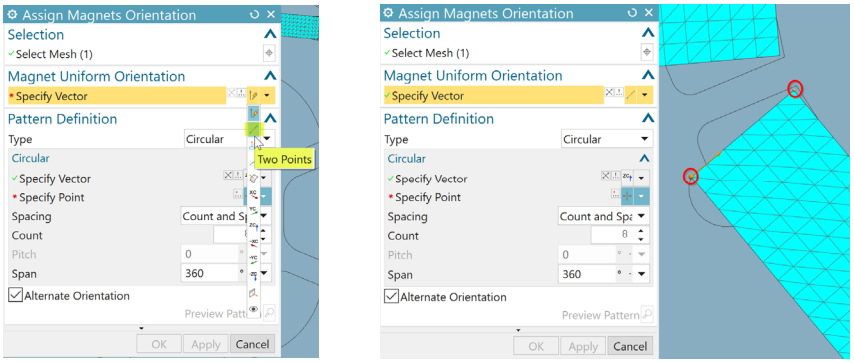

Figure 46: Specify the direction vector

- Set pattern type to Circular, using the Point Dialog method to set origin as Specify Point. Select Count and Span for Spacing, then set Count to 8, Span to 360, and check the option Alternate Orientation.

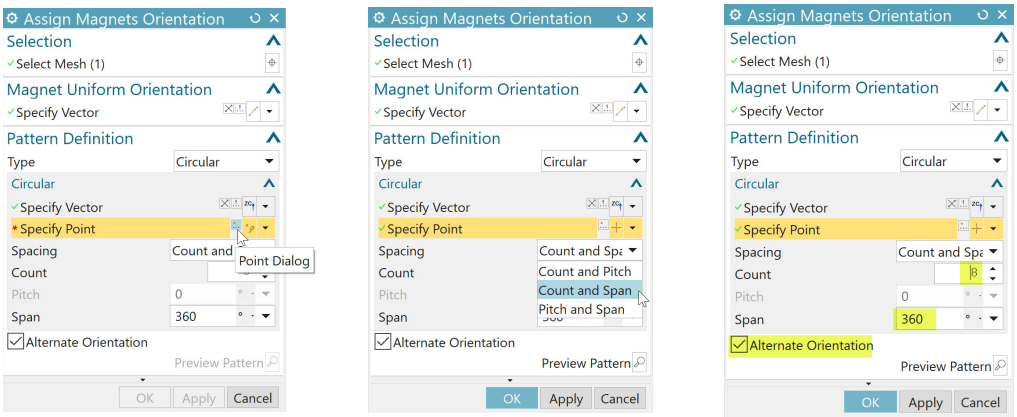

Figure 47: Specify the pattern settings

- Click Preview Pattern to show the direction assignment. Set Specify Vector to ZC and click Apply if the direction is ok.

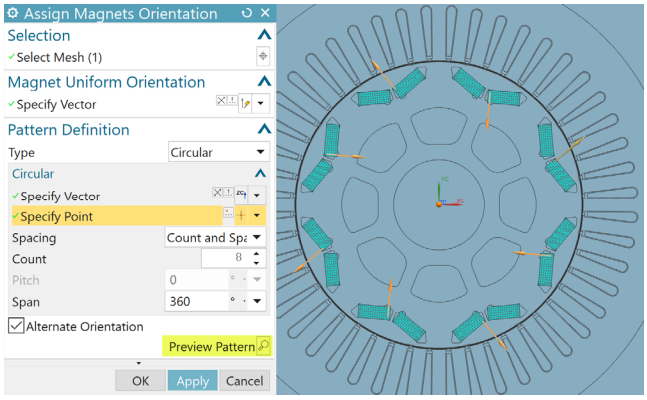

Figure 48: Magnets orientation preview

- Clicking Apply instaed of OK will keep the Assign Magnets Orientation window open with the same settings. Select another magnet mesh, as shown below.

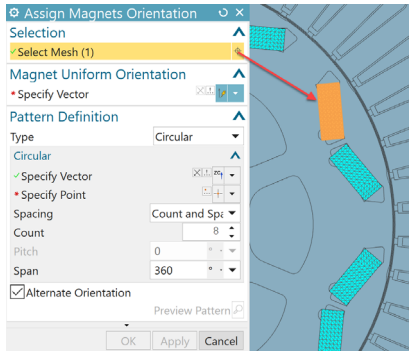

Figure 49: Select another mesh for assigning direction

- Use the Two Points method for specifying the vector, and select the two vertex of the inner edge as shown below

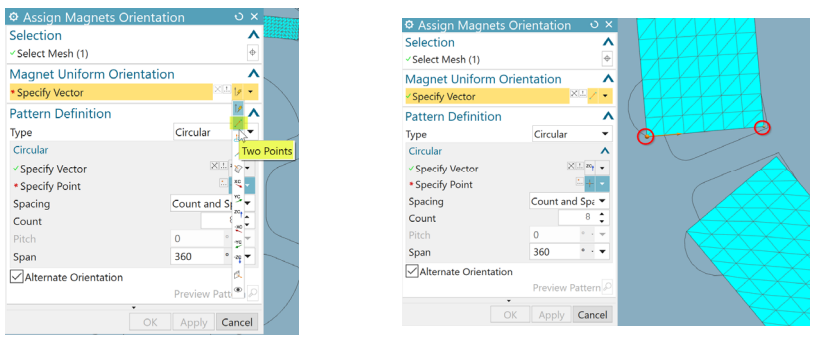

Figure 50: Specify the direction vector

- Use Point Dialog method to set origin as Specify Point. Then keep all other settings and click OK. To display the magnets direction, click Display Magnets on the Low Frequency EM ribbon

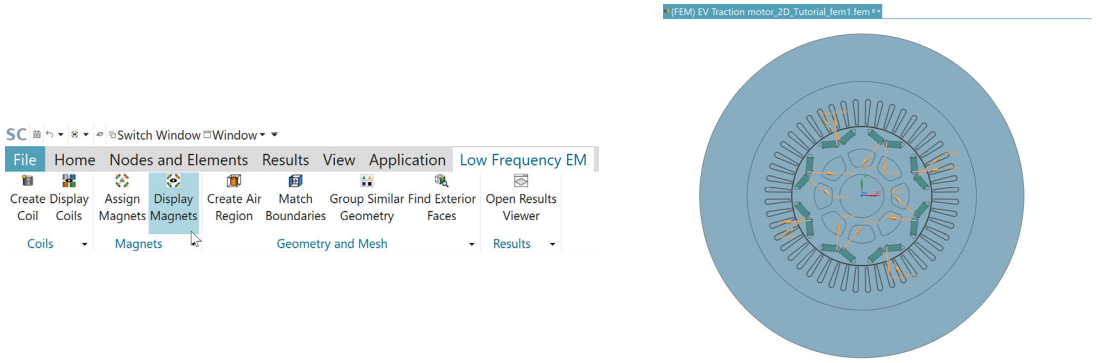

Figure 51: Display the direction of magnets

- Click File|Save|Save all to save the files.

4. Simulation Set-Up

The simulation set-up involves creating coils, circuits, motion objects and assigning boundary conditions, which can only be performed in a sim file.

- Right-click "*EV Traction motor_2D_Tutorial_fem1.fem*" in the Simulation Navigator, select Display Simulation and then "*EV Traction motor_2D_Tutorial_sim1.sim*"; this makes the sim the work part.

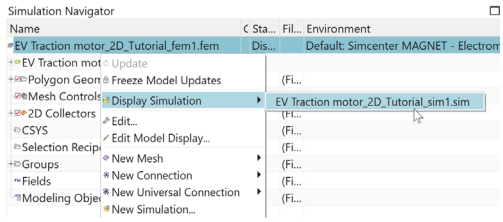

Figure 52: Make sim the work part

4.1. Create Coils

Coils are implemented as simulation objects and are associated to a solution. Before creating coils, at least one solution should be created.

- Right-click "*EV Traction motor_2D_Tutorial_sim1.sim*" in the Simulation Navigator, and then select New Solution.

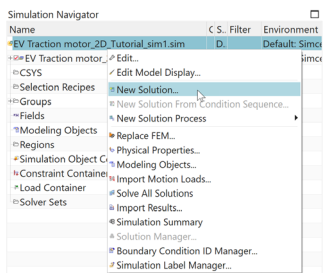

Figure 53: Make sim the work part

- In the Solution window, set Solution Type to Transient.
- In Time Steps, set Stop Time=1 s and Time Step=0.1 s.
- Click OK; the new solution is listed in the Simulation Navigator.

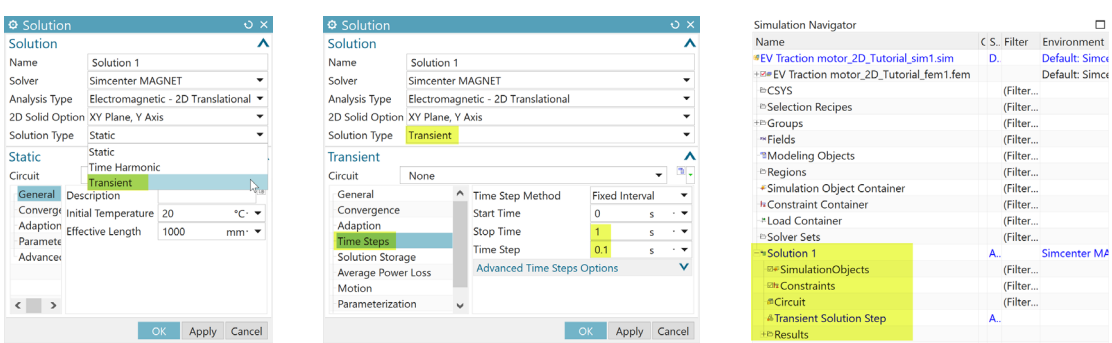

Figure 54: Create a new solution

- Since a 2D coil is defined using a face coil, the polygon body has to be visible. Unchecking the 2D Collectors to hide the meshes will provide a better view for this procedure.

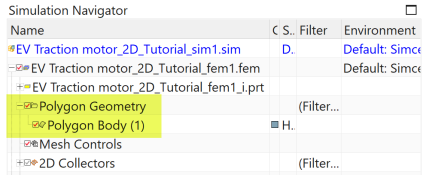

Figure 55: Make Polygon Body visible

- From the Low Frequency EM ribbon, under the Coils command group, click Create Coil.

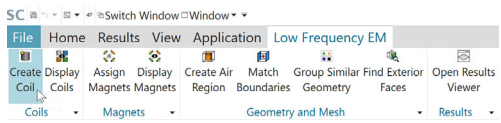

Figure 56: Launch Create Coil tool

- In the Coil window, first set the Name to Coil and then select Entry face and Exit Face, as shown below.

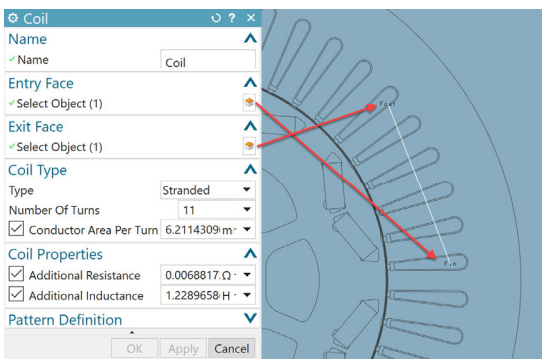

Figure 57: Select coil terminal faces

- In the Coil Type section of the Coil window, select Stranded as Type, set Number of Turns=11, check Conductor Area Per Turn and set its value as 6.211430903136 mm^2. In the Coil Properties section, check Additional Resistance and Additional Inductance and set their values as 0.006881722557661 ohm and 1.228965860695e-05 H, respectively.

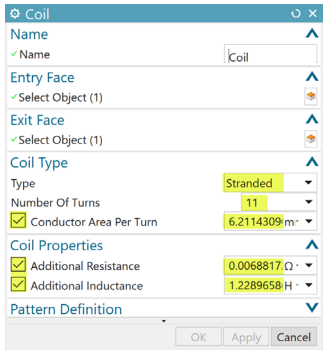

Figure 58: Set Coil type and properties

- If required, expand the Pattern Definition section, then select Circular as Type and set origin as Specify Point.
- Select Count and Span as Spacing method, set Count=24 and Span=360.
- Click Preview Pattern to show all coils.

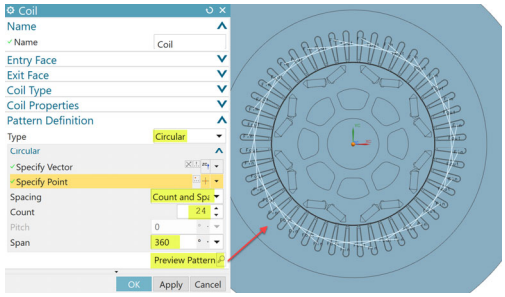

Figure 59: Set Coil pattern definition

- Click OK to finish the coil definition. In the Simulation Navigator, a folder labelled Coil-Pattern will be listed under the Simulation Object Container, which includes the 24 coils. To display the coils, click Display Coils from the Low Frequency EM ribbon.

  Note: The coils are active and automatically added to the active solution that was created earlier.

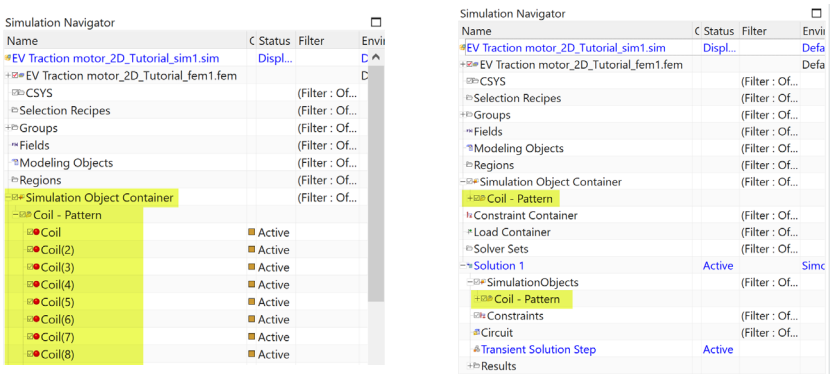

Figure 60: Coil-Pattern folder

- Click File|Save|Save all to save the files.

4.2. Build the Circuit

To group the coils into three phases and then set up the excitation, a circuit is built for the corresponding solution.

- Under the active solution, right-click Circuit and then select New or Replace Circuit and Circuit-Transient.

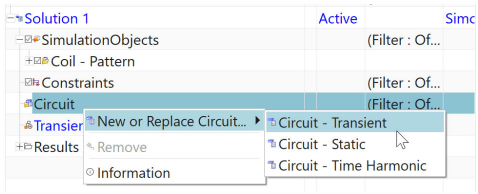

Figure 61: Launch Circuit editor

- In the Circuit-Transient window, click Edit Circuit to open the Circuit Editor. All the circuit components are listed in the left panel of the circuit editor.

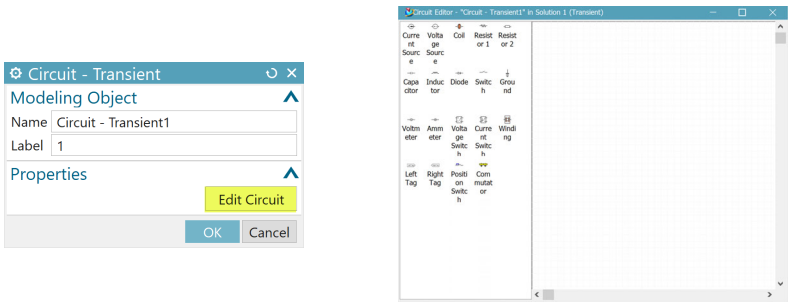

Figure 62: Launch Circuit editor

- A Winding is a circuit component for grouping coils. Drag and Drop a Winding from the left panel into the editor area; a Winding Properties window appears. The coils created earlier are all listed in Available Coils.

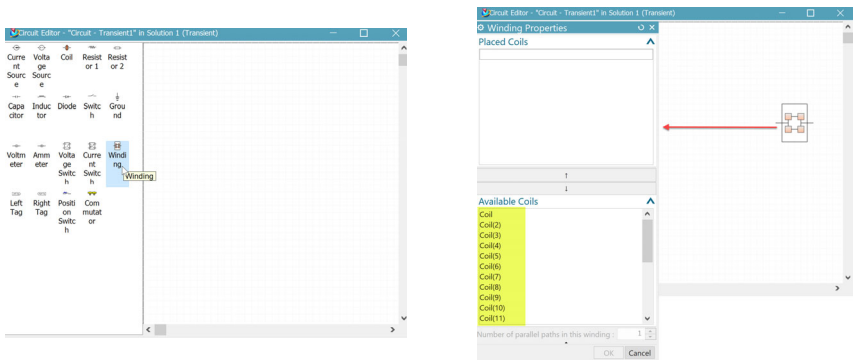

Figure 63: Create a winding

- Hold the Ctrl key and select Coil, Coil(4), Coil(7), Coil(10), Coil(13), Coil(16), Coil(19) and Coil(22) from the Available Coils list and click the up arrow to add them into Placed Coils. Since all the selected coils are connected serially, set Number of parallel paths in this winding to 1 (**Note**: keep this setting for Winding2 and Winding3).

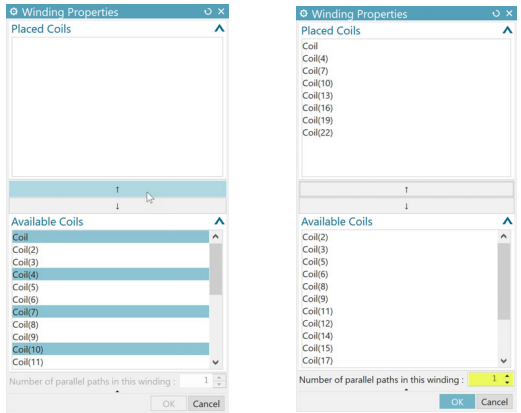

Figure 64: Create a winding

- Click OK to finish the first winding (i.e. Winding1). Follow the same steps to create Winding2 and Winding3 by grouping the coils, as shown below:
  - Winding2: Coil(3), Coil(6), Coil(9), Coil(12), Coil(15), Coil(18), Coil(21) and Coil(24)
  - Winding3: Coil(2), Coil(5), Coil(8), Coil(11), Coil(14), Coil(17), Coil(20) and Coil(23)

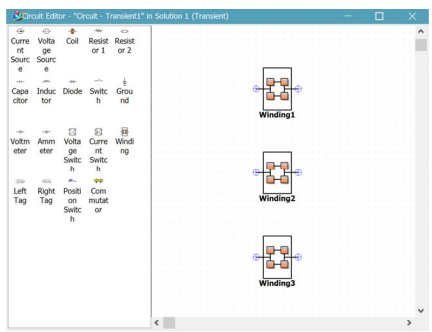

Figure 65: Winding1,2 and 3

- To edit the winding connection, right-click Winding1 and select Properties. Click Edit Winding Circuit in Winding Properties window.

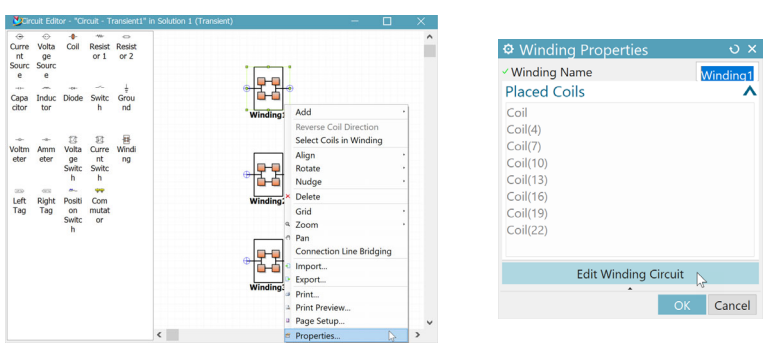

Figure 66: Properties of Winding1

- In the Edit Winding Circuit window, first hold the Ctrl key and select Coil(4), Coil(10), Coil(16) and Coil(22), then right-click and select Reverse Coil Direction.
- Close the edit window and click OK to finish the editing.

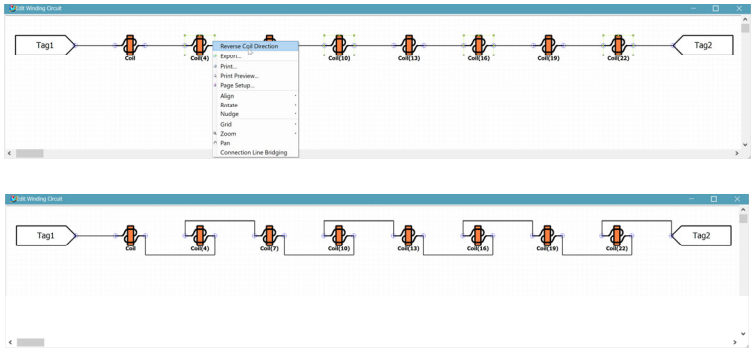

Figure 67: Edit Winding1 circuit

- Following the same steps, edit Winding2 and Winding3 by reversing the following coil connections:
  - Winding2: Coil(6), Coil(12), Coil(18) and Coil(24); right-click and select Reverse Coil Direction.
  - Winding3: Coil(2), Coil(8), Coil(14), Coil(20); right-click and select Reverse Coil Direction.

Figure 68: Edit Winding2 and Winding3 circuits

- Drag and Drop three Current source components into the editor area.

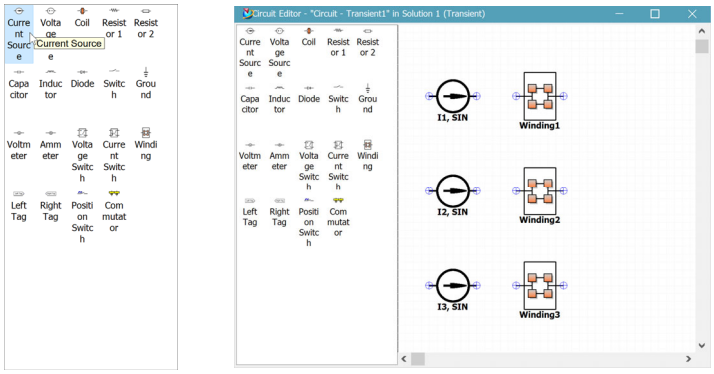

Figure 69: Current source I1, I2 and I3

To edit the source, right-click the source component and select Properties. Since the first solution is to calculate the back-EMF at a no load condition, just leave the default source settings.

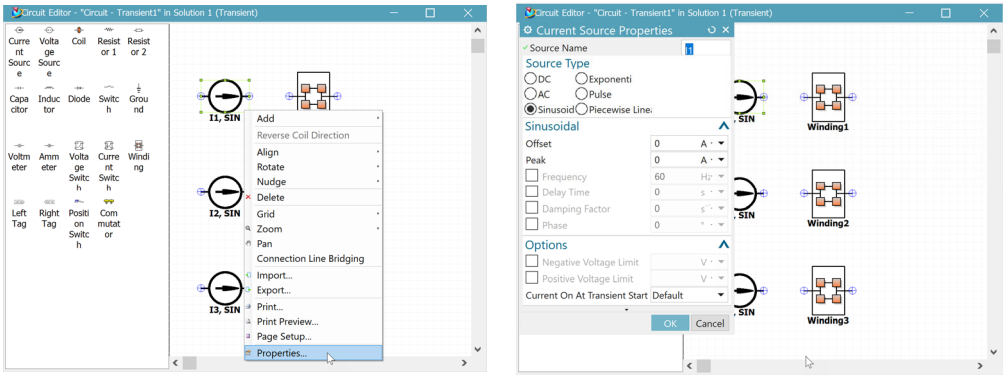

Figure 70: Properties of Current source

Hover the mouse over the right terminal of I1, when an x is shown on the tip of the mouse arrow, right-click the mouse, then a rectangular is shown. Drag the mouse to the left terminal of Winding1, when a rectangular is shown, right-click the mouse to finish the connection.

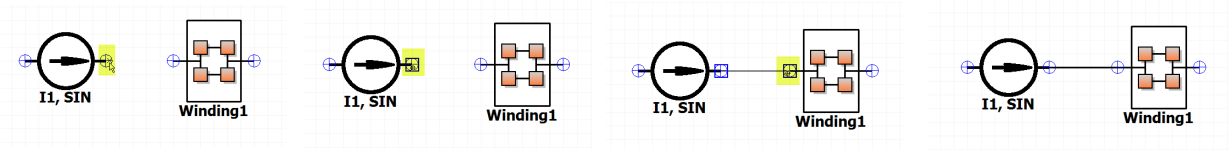

Figure 71: Circuit connection

Follow the same steps to finish the connection as shown below. Close the circuit editor window and click OK in the Circuit-Transient window to finish the circuit editing. Circuit-Transient1 is shown under Circuit of Solution 1.

Note: The circuit is stored as a Modeling Object.

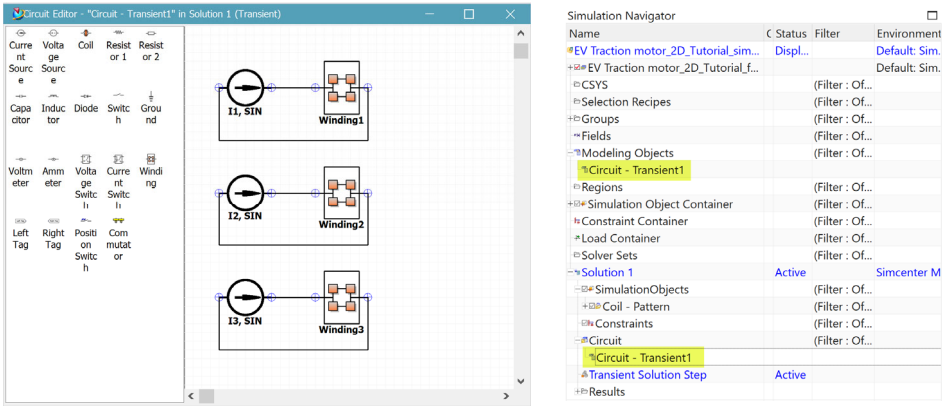

Figure 72: Final circuit

Click File|Save|Save all to save the files.

4.3. Setup Motion Component

The motion component is implemented as a Simulation object and associated to the Transient analysis type.

- Click Motion Component from Home|Loads and Conditions commands group. The Motion Component window appears.

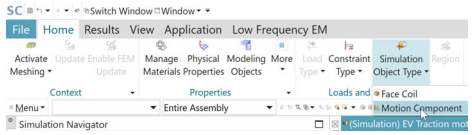

Figure 73: Motion Component command

- Select Rotary Component type at the top cell. Make sure the selection method is set to Adjacent Faces, go to the Components section and click the rotor core face; all the adjacent faces (75, in all) are selected.

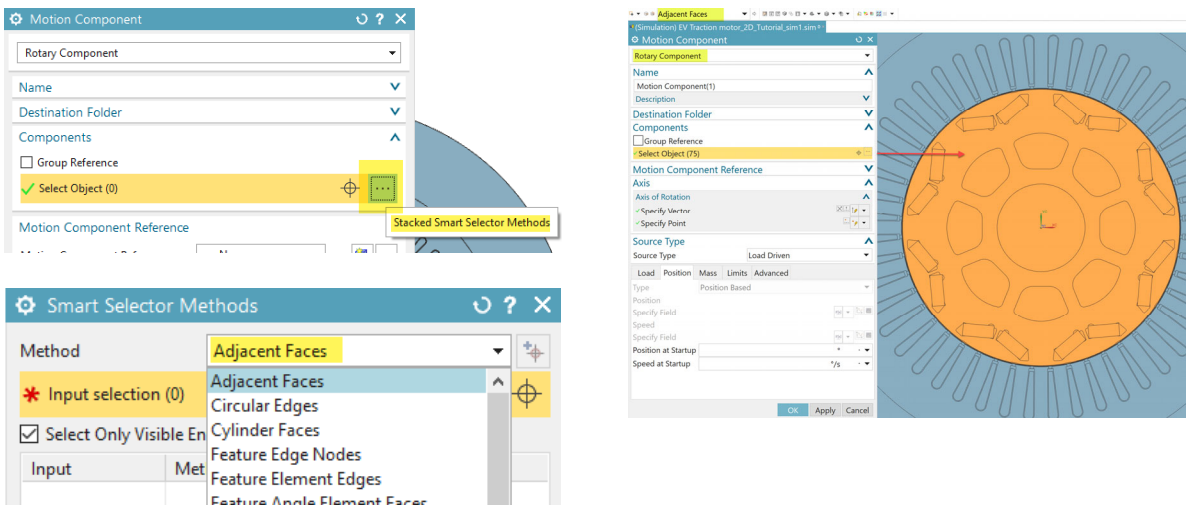

Figure 74: Select object for Motion Component

- Change the selection method to No method, zoom in and select the remesh face on the rotor side, increasing the Select Object total to 76. Now, all the objects for the motion component are selected.

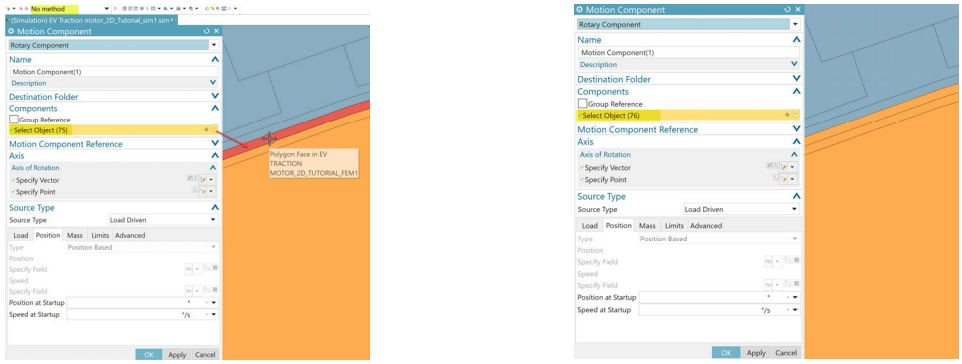

Figure 75: All objects for Motion Component

- In the Axis section, select ZC from the Specify Vector drop-down list.
- Use Point Dialog to set origin to Specify Point.

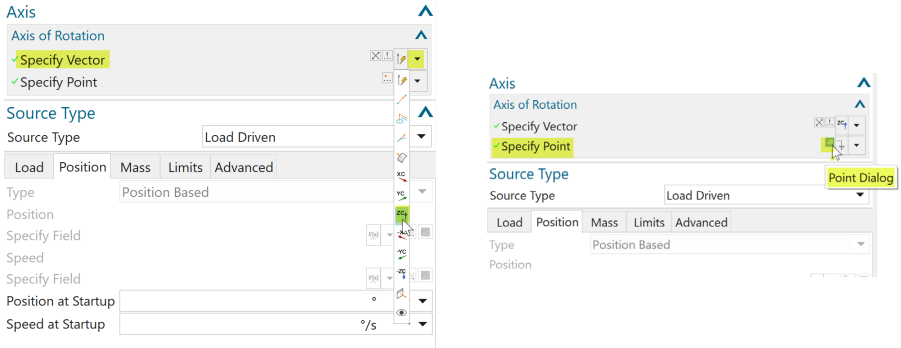

Figure 76: Set Axis of rotation for Motion Component

- In Source Type section, select Velocity driven.
- Under Position tab, for Type, select Speed Based.

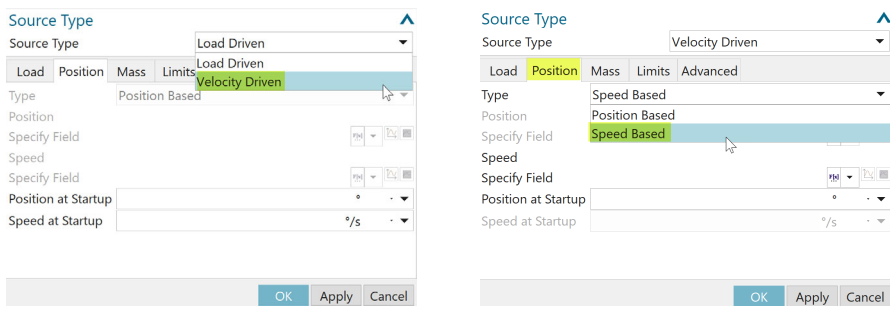

Figure 77: Set Source type and position type for Motion Component

- Select Table Constructor from the Specify Field drop-down list.
- Click Next to accept all defaults until the Definition table appears.
- In the Table Field/Definition table, type 0 for "time(s)" and 18000 for "angular velocity".
- Click OK to finish the setting.
- Note: The simulation calculates the back-EMF and torque at rated speed which is 3000RPM (18000°/s)

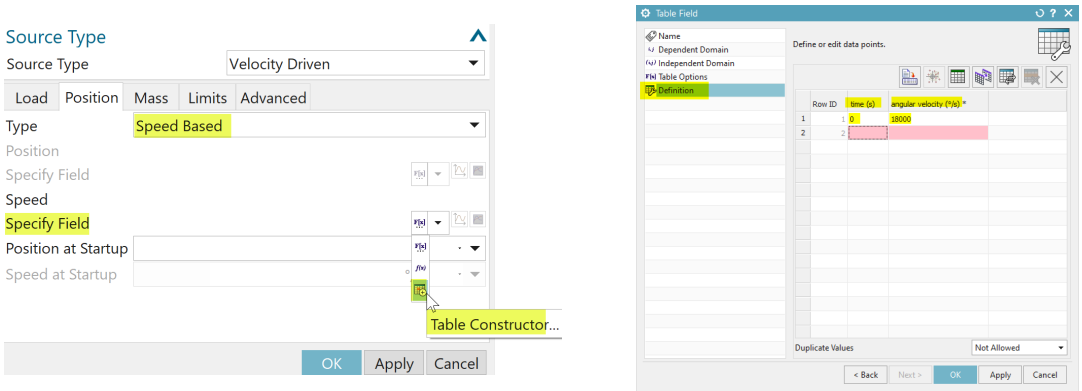

Figure 78: Specify speed for Motion Component

- Click OK in Motion Component window to finish the motion setup. In the Simulation Navigator, the motion component is listed under Simulation Object Container and is automatically added to the active solution.

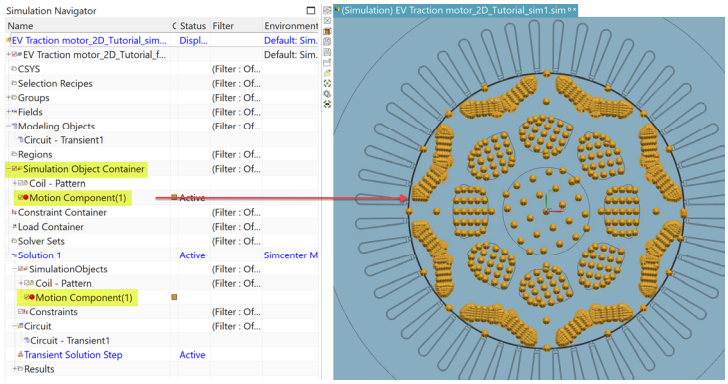

Figure 79: Motion Component

- Click File|Save|Save all to save the files.

4.4. Setup solving options

The solving related options are set up by editing the solution.

- Right-click Solution 1 and select Edit.

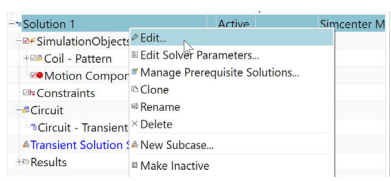

Figure 80: Edit Solution 1

- In the Solution window, change the solution Name to Solution 1-Back-EMF at 3000RPM.
- In General tab, set Effective Length=50.8 mm
- Note: The setting Effective Length is specific to 2D and used for the global results quantities calculation. In this case, the stack length of the motor is 50.8mm.

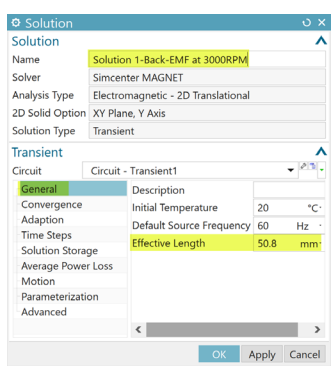

Figure 81: Edit solution name and effective length

- In Convergence tab, change Polynomial Order to 2 (if required). Accept all the other settings.

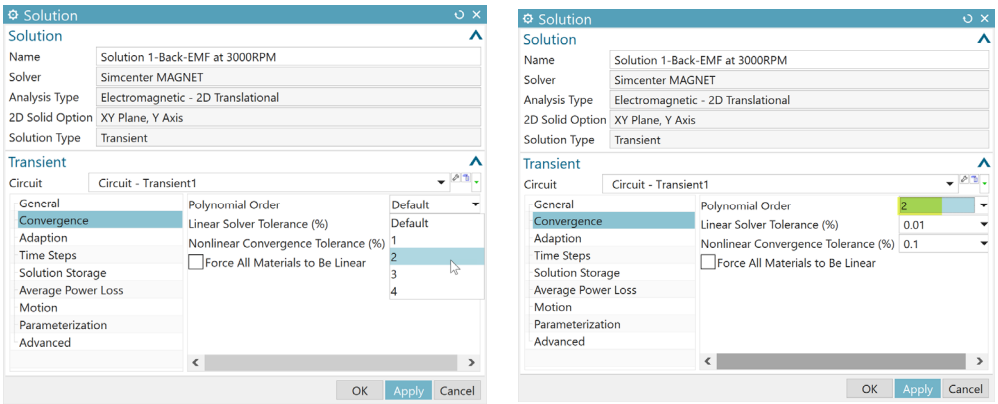

Figure 82: Edit Convergence settings

- In Time Steps tab, change unit to ms and set Stop Time=5 ms and Time Step=0.05 ms. Accept all the other settings.
- Note: Since the motor has 8 poles and the rated speed is 3000 RPM, the electrical frequency is 200 Hz (f=p*n/120, where **p** is the number of poles and **n** is the speed). For one electrical cycle, this corresponds to 5 ms. This time step setting makes the simulation run 100 steps in one electrical cycle.

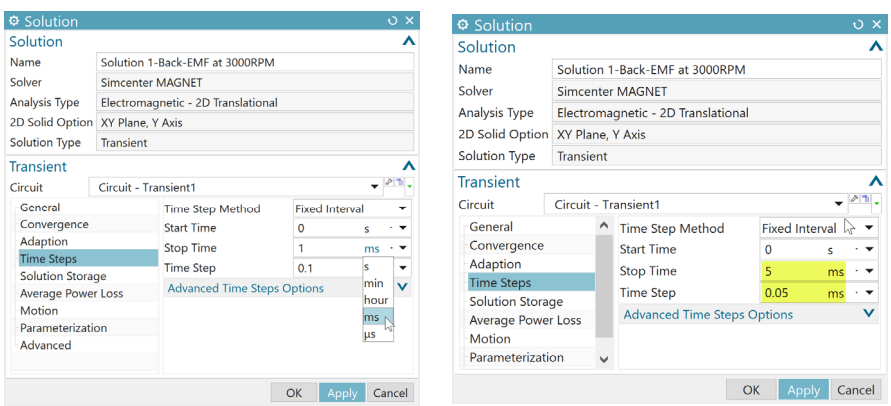

Figure 83: Edit Time Steps settings

- Keep the default settings for the other tabs and click OK to finish the solution editing.
- Click File|Save|Save all to save the files.

4.5. Clone solution

Solution 1 was created to calculate the back-EMF at rated speed. Now, to calculate the torque at rated condition, another solution needs to be set up. Since it is almost the same as the one for calculating back-EMF, except for the current source settings, the solution can be cloned from Solution 1, with just the source settings being modified. The clone will keep the same settings as the source solution.

- Right-click Solution 1-Back-EMF at 3000RPM and select Clone. A new solution is created, as shown below

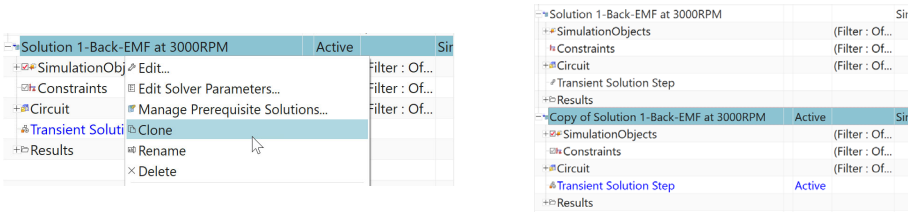

Figure 84: Clone a solution

- Right-click Copy Solution 1-Back-EMF at 3000RPM and select Edit. Change the solution name to Solution 2-Torque at 3000RPM and click OK

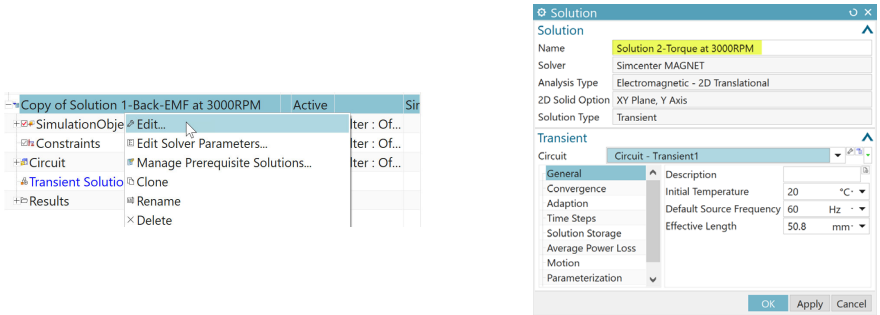

Figure 85: Edit solution 2

- Right-click Circuit – Transient1 under Circuit of Solution 2-Torque at 3000RPM and select Remove. This will remove the circuit from the solution. Then right-click Circuit, select New or Replace Circuit and then Circuit-Transient.

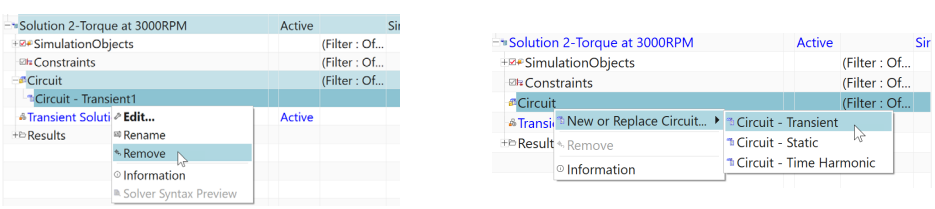

Figure 86: Create a transient circuit for solution 2

In the Circuit-Transient window, click Edit Circuit. Note that the name is now Circuit – Transient 2,

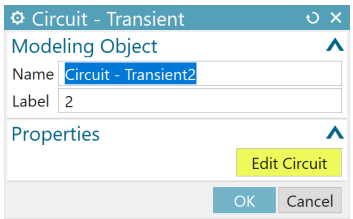

Figure 87: Launch Circuit editor for solution 2

In the Circuit Editor window, right-click in the edit area and select Import. Select the Circuit – Transient1 and click OK in the Import Circuit window.

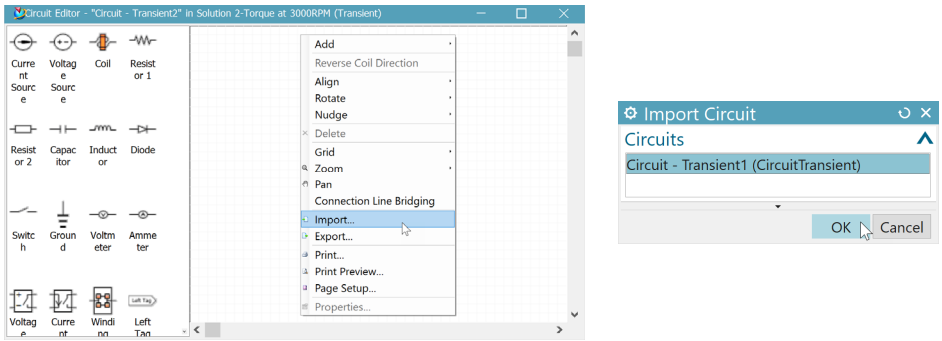

Figure 88: Import a circuit

There will be a warning message popup, click Yes. The circuit is imported, as shown below.

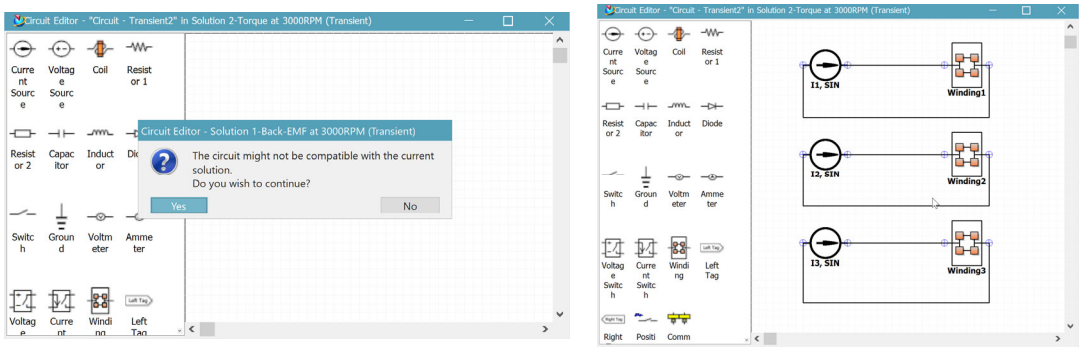

Figure 89: Finish Import circuit

- Right-click current source I1, SIN and select Properties. In the Current Source Properties window, set Peak=150 A, Frequency=200 Hz, Delay Time=0 s, Damping Factor=0 s^{-1} and Phase=230°.
- Click OK.

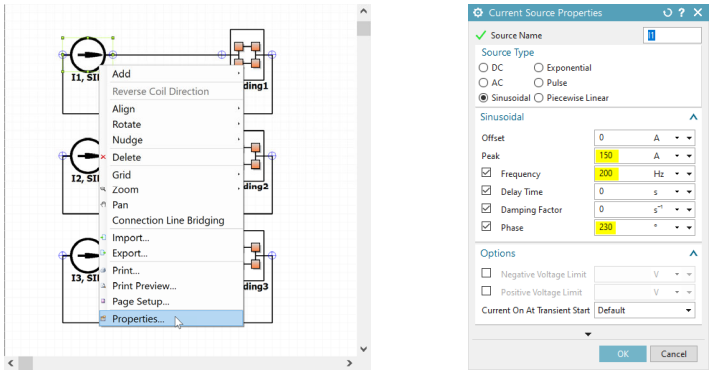

Figure 90: Edit the source I1 settings

- Follow the same steps to set current source I2, SIN and I3, SIN, as shown below:
  - I2, SIN: Peak=150 A, Frequency=200 Hz, Delay Time=0 s, Damping Factor=0 s^{-1} and Phase=110°.
  - I3, SIN: Peak=150 A, Frequency=200 Hz, Delay Time=0 s, Damping Factor=0 s^{-1} and Phase=350°.

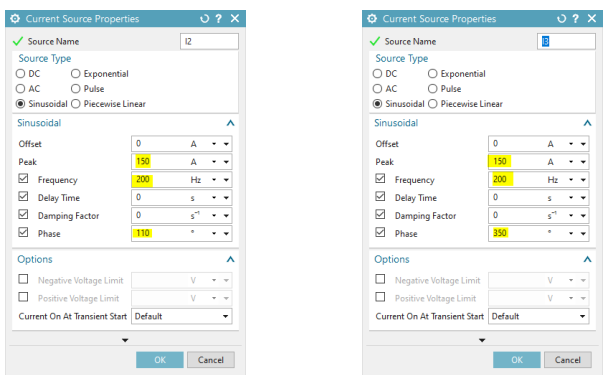

Figure 91: I2, I3 Source settings

- Close the Circuit editor and click OK in the Circuit-Transient window. There are now two circuits in the sim file, as shown below

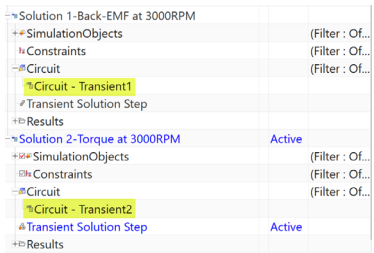

Figure 92: Two solutions with two circuits

- Click File|Save|Save all to save the files.

4.6. Solve

There are different ways to launch the solving process.

- Right-click EV Traction motor_2D_Tutorial_sim1.sim and select Solve All Solutions. In the popup window, check the option Simultaneous and then click OK. This will launch two solves simultaneously.

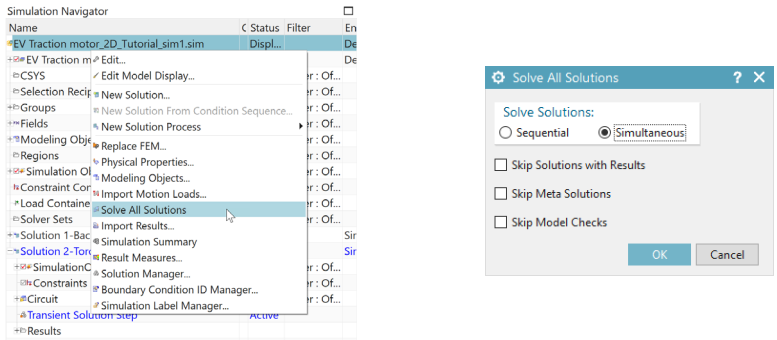

Figure 93: Launch simultaneous solves

- Or, right-click the active solution, and select Solve. In the popup window, click OK to start the solve

  Note: Clicking Edit Solver Parameters and unchecking the option Run Solver in Foreground allows you to continue working on the sim during the solving process.

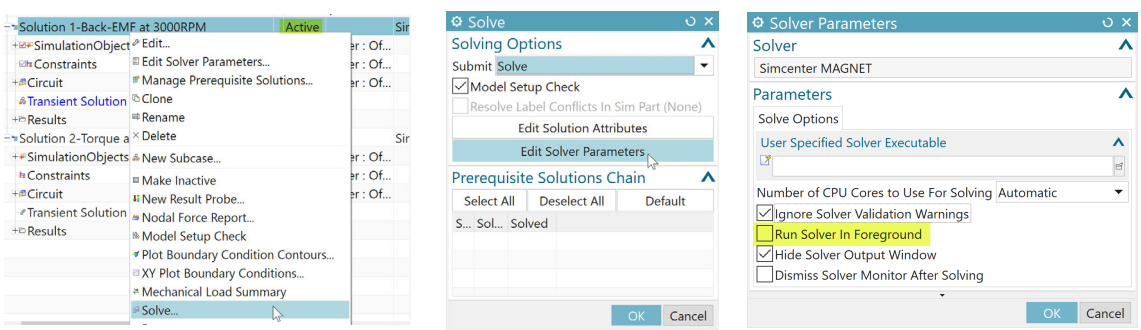

Figure 94: Launch active solution

Once a solve is started, a solver monitor window appears, showing the solving progress.

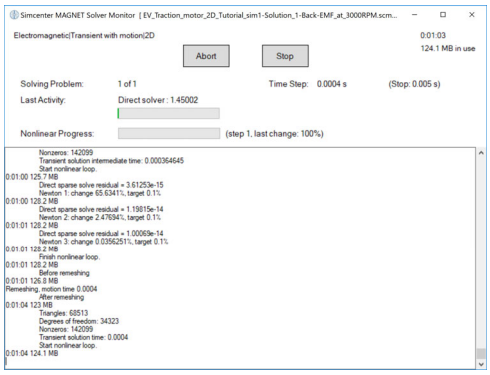

Figure 95: Solver monitor

When the solve is finished, a button, Start MAGNET Results Viewer, is available for launching the Low Frequency EM – Results Viewer.

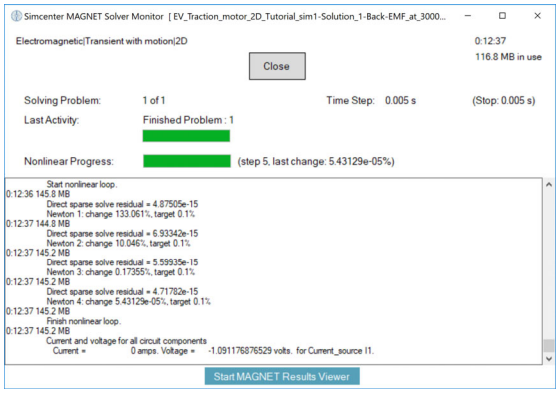

Figure 96: Finished Solve

5. Results

The post-processing of results is available through a separate tool (i.e. Low Frequency EM - Results Viewer) that can be launched directly from Simcenter.

There are a couple of ways to launch the results viewer.

- Click Start MAGNET Results Viewer at the bottom of the solver monitor.

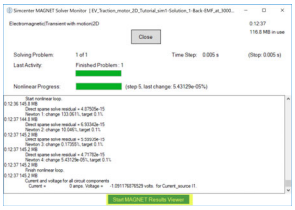

Figure 97: Launch Results Viewer from Solver Monitor

- Click Open Results Viewer from the Low Frequency EM ribbon. This will open the results of the active solution.

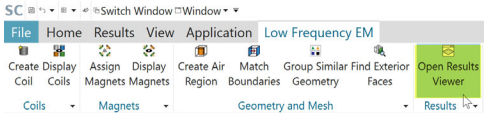

Figure 98: Launch Results Viewer from Low Frequency EM ribbon

5.1. Results for Back-EMF solve

Note In version 2020.1 of the Results Viewer:

- **TreeView** panel has been renamed **Datasets and Solutions**.
- **Component** has been renamed **Object**.

  Functionality has not changed.

- After finishing the back-EMF solve, click Start MAGNET Results Viewer at the bottom of the solver monitor. The results viewer will open in a separate window. In Datasets and Solutions, the default datasets are listed: Solution mesh, plus the B and Flux function fields.

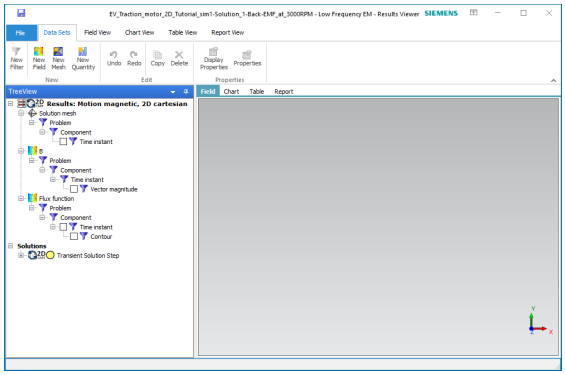

Figure 99: Low Frequency EM Results viewer

- Check the Vector magnitude and Contour filters for the B and Flux function fields, respectively. The shaded plot is displayed in the Field view window.

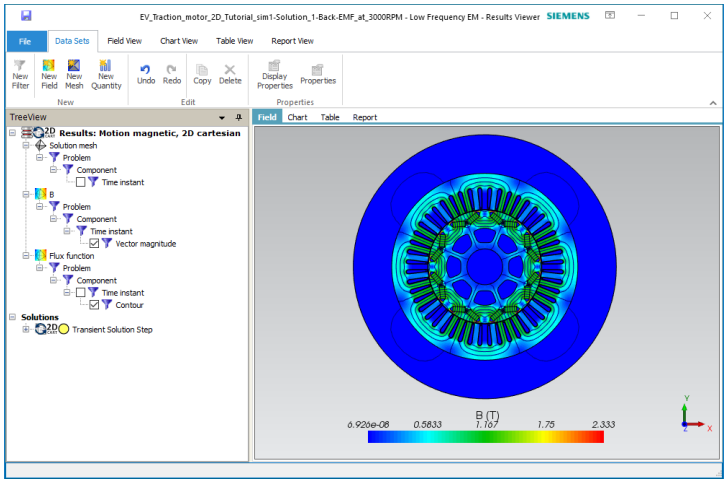

Figure 100: B field and flux function field view

- To probe the air gap flux density, right-click the Vector magnitude filter, select New Filter and then Arc probe. Right-click the Arc probe filter and then select Properties. In the 'Arc probe' Filter Properties page, set Start point= 80.575, 0, 0 and Arc angle=360.
- Accept all other defaults and click OK.

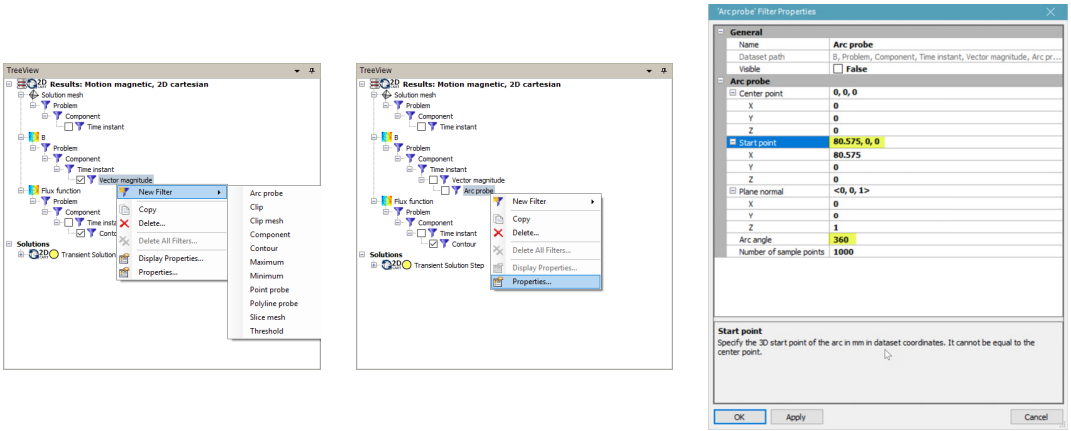

Figure 101: Create arc probe for air gap flux density

- Check the Arc probe filter and click the Chart tab; it will display the flux density plot along the air gap as shown below.

  Note: The air gap flux density is around 0.85 T, which is within the common design requirement for rare-earth PM motor.

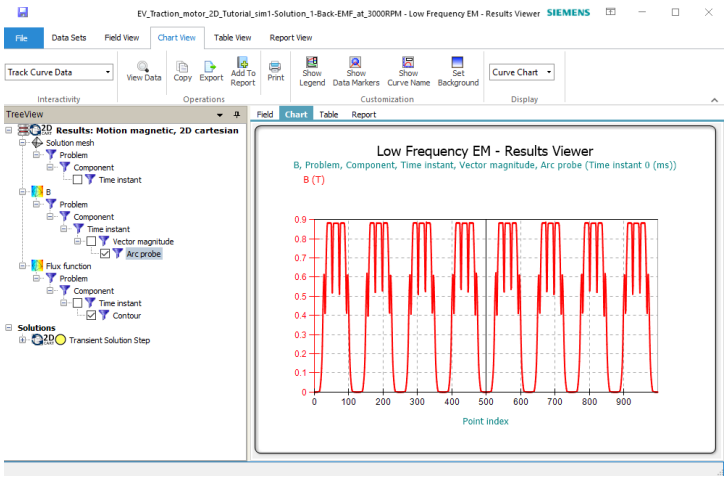

Figure 102: Voltage plot of three windings

- To plot the back-EMF, right-click Results, select New Quantity and then Voltage.

The Voltage quantity will list in the Datasets and Solutions.

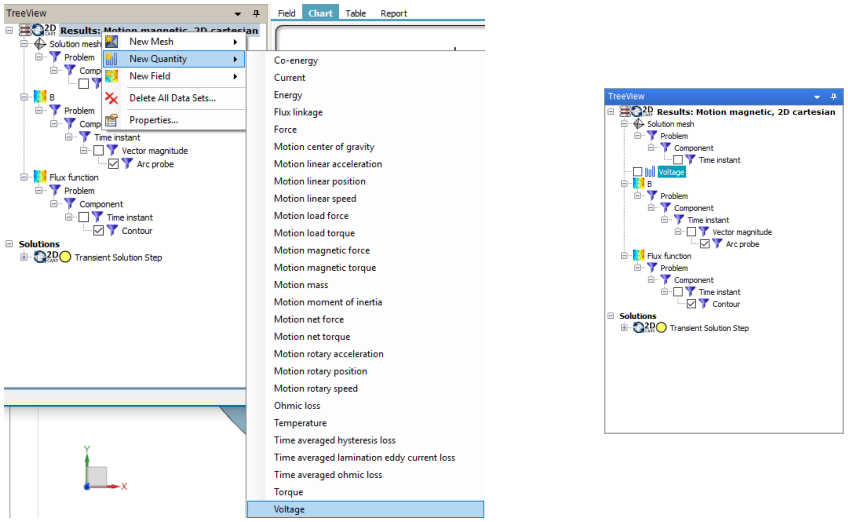

Figure 103: Create new result quantity: Voltage

- Right-click the Voltage quantity, select New Filter and then Object. Right-click the Object filter and select Properties. In the Object Filter Properties page, set Selection mode to Include, and select Winding1, Winding2 and Winding3 from the drop-down list of Selected Objects.
- Click OK.

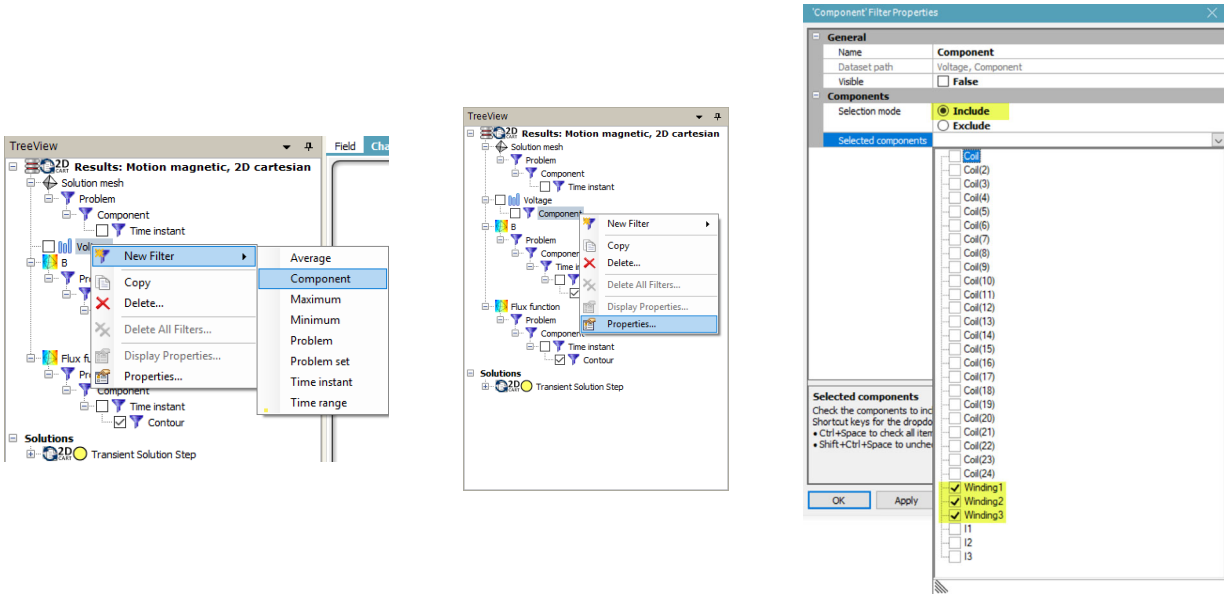

Figure 104: Create new filters for result quantity: Voltage

- First, uncheck the Arc probe filter, then check the Object filter and click Chart tab; it will display the voltage plot of three windings as shown below.

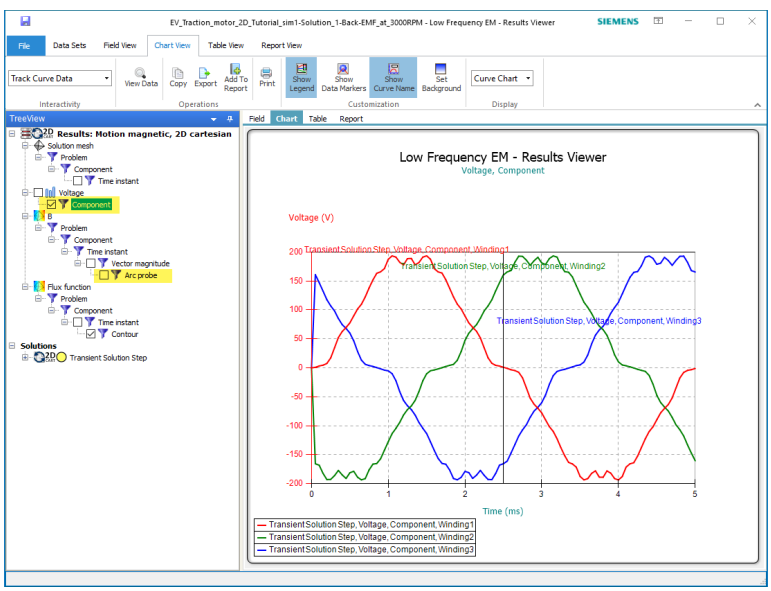

Figure 105: Voltage plot of three windings

5.2. Results for Torque solve

- The same procedure can be used to view the results of Solution 2 for torque solve. From the Simulation Navigator, make “Solution 2-Torque at 3000RPM” the active solution. Click Open Results Viewer from the Low Frequency EM ribbon. The results viewer will open in a separate window.
- Right-click Object filter under the B field and select Properties. In the ’Object’ Filter Properties page, set Selection mode to Exclude, and select Stator air/2d_mesh(167) from the drop-down list of Selected Objects.
- Click OK.

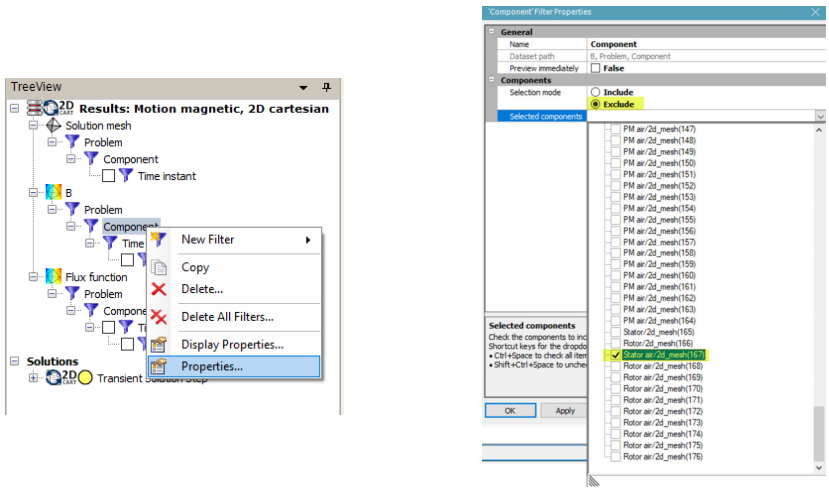

Figure 106: Set Object filter to exclude the stator air

- Check the Vector magnitude filter for the B field. The shaded plot is displayed in the view window.

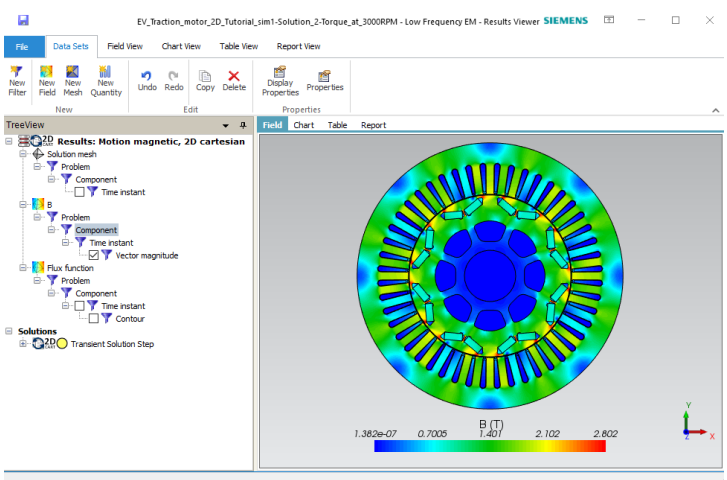

Figure 107: B field at rated condition

- Right-click Vector magnitude and select Display Properties. In the Display Properties page's Data range section, set Range type to Manual, and Minimum=0, Maximum=2.
- Click OK.

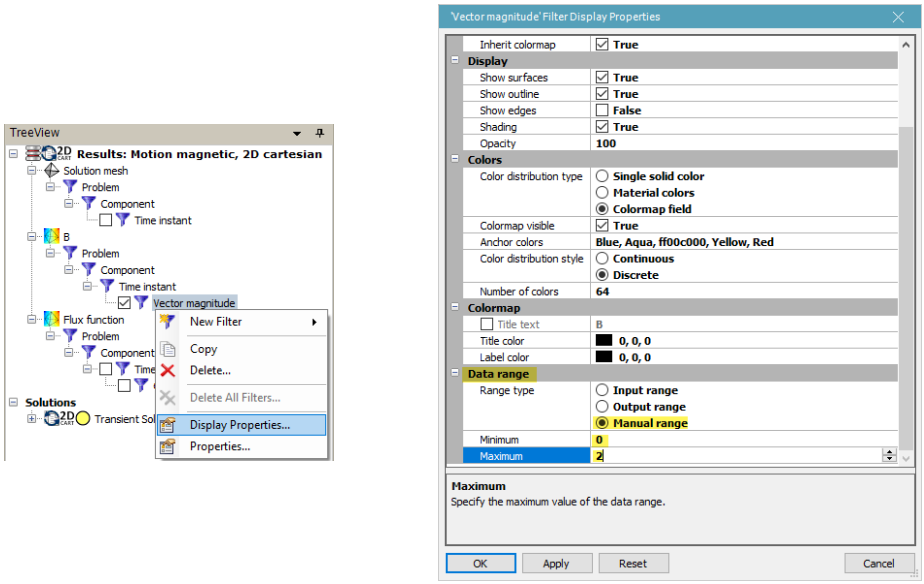

Figure 108: Set display properties for B field

- The B field will automatically update, as shown below, revealing that the motor is heavily saturated at the rated condition.

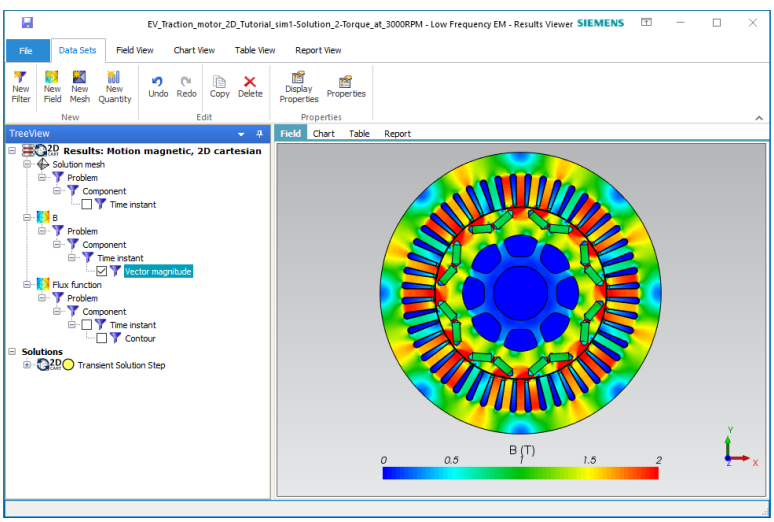

Figure 109: B field at rated condition

To view the torque waveform, right-click Results, select New Quantity and then Motion magnetic torque.

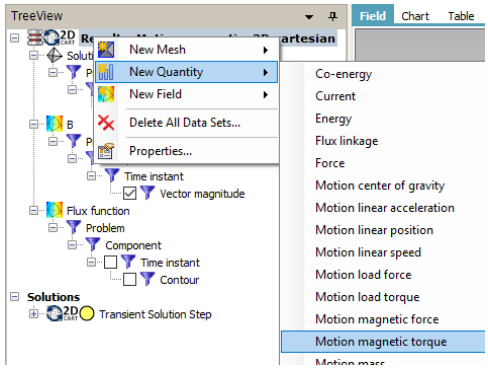

Figure 110: Create new filters for result quantity: Motion magnetic torque

Check the Motion magnetic torque quantity and click Chart tab; it will display the torque plot, as shown below. Due to the saturation, the torque performance is lower than the expected value.

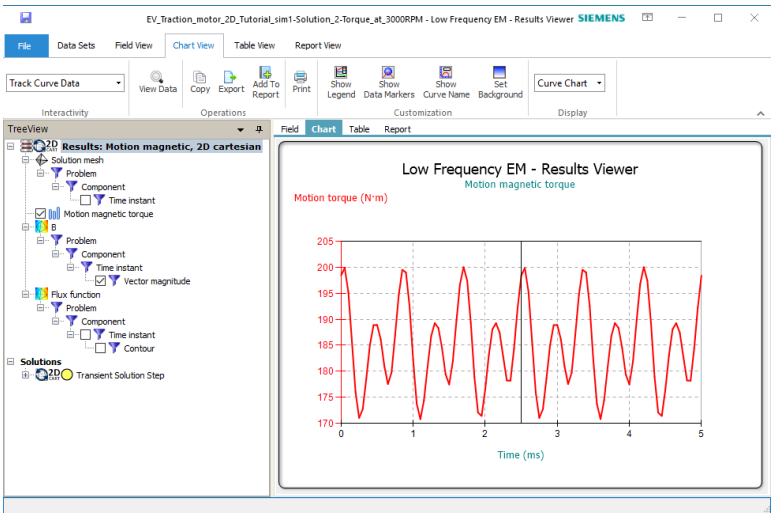

Figure 111: Torque plot at rated condition