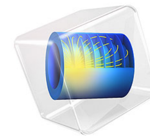
Created in COMSOL Multiphysics 6.2



Permanent Magnet Motor with Campbell Diagram

Introduction

In this modeling example, noise emitted from an electrical motor due to electromagnetic forces is studied. A simplified motor structure is first analyzed to find its natural frequencies and eigenmodes. Following this a time periodic electromagnetic simulation is performed to establish the magnitude and frequency of electromagnetic forces occurring in the motor. Finally the electromagnetic force harmonics are applied as load to determine the resulting structural vibration and surrounding acoustical emission of the motor. From this a Campbell diagram can be generated that provides insight into which harmonics of the electromagnetic forces are causing the most prominent noise.

Model Definition

The focus of this example is on demonstrating the most straightforward way of coupling electromagnetic forces with an acoustic-structural model and postprocessing relevant results. As such a very simple 2D geometry is used for all physics to minimize the level of complexity not directly related to the multiphysics coupling.

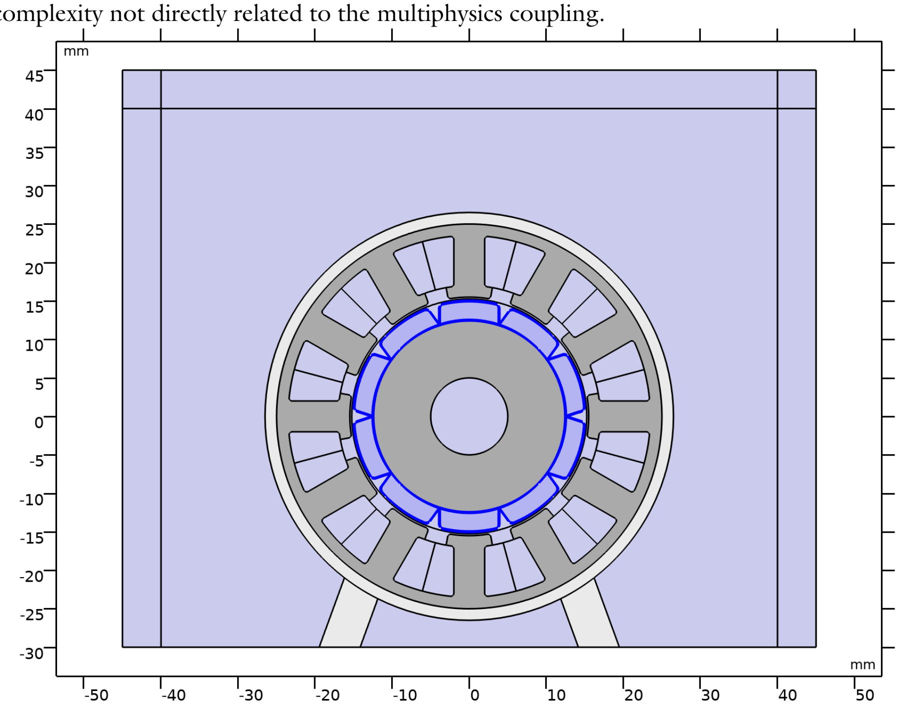


Figure 1: A simple geometry of an electrical motor structure and near surrounding space.

For a more practical case it is normally necessary to model the motor structure and acoustical environment in 3D while electromagnetically a 2D representation can still cover many cases. In such a configuration the main difference from this example would be that the electromagnetic forces are extruded into 3D space with an extrusion coupling operator.

STRUCTURAL CONSIDERATIONS

The Solid Mechanics interface is applied to the stator core, motor housing and foot mounts only.

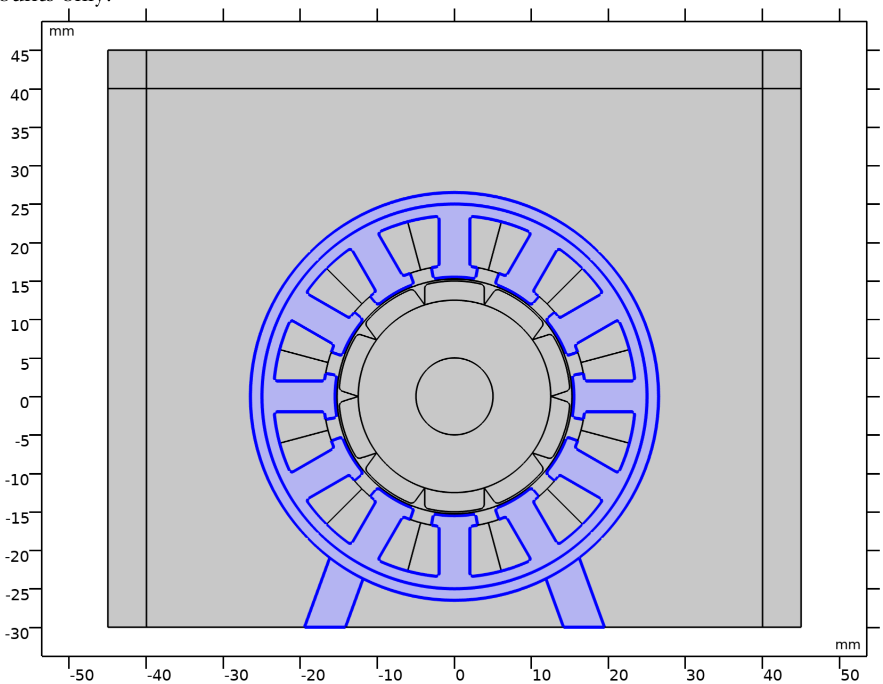


Figure 2: Solid Mechanics domains.

The lower horizontal boundaries of foot mounts are constrained with zero displacement. Electromagnetic forces are applied to all interior boundaries of the stator core.

.

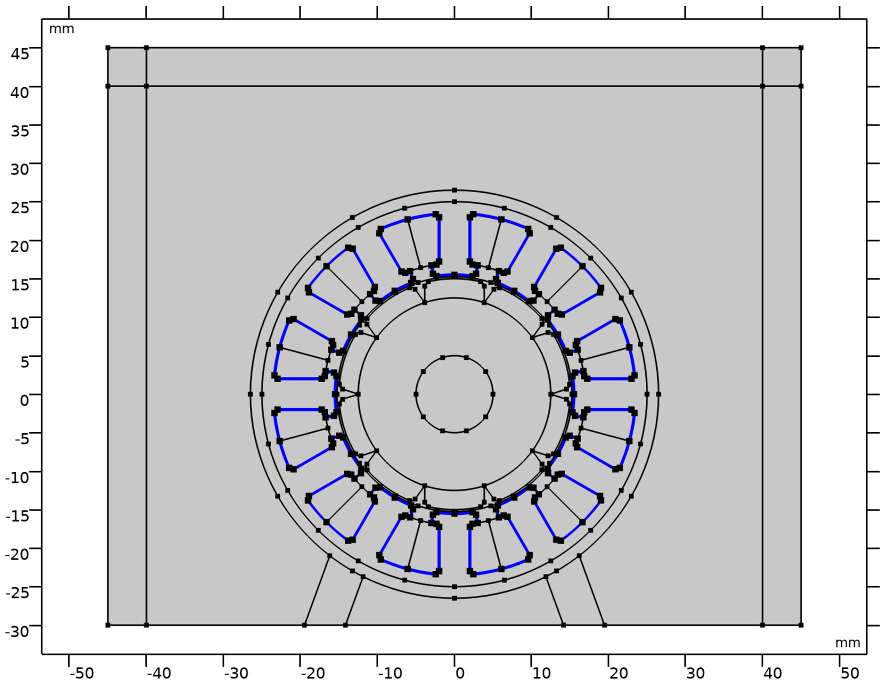


Figure 3: Boundaries at which the electromagnetic forces are applied.

The harmonics of a distributed magnetic force output by the Magnetic Machinery Time Periodic interface can be accessed by calling the Fourier component of the Maxwell surface stress tensor in its operator form. As an example, the x-component can be specified as **mmtp.nTX_1_fft(freq/f_el)**, where **freq** is the swept variable of the frequency domain study and **f_el** is the fundamental electrical frequency. When **freq** is swept over **n** values, all being a multiple of **f_el**, the ratio **freq/f_el** refers to 1st, 2nd, 3rd, …, **n**th Fourier component of the Maxwell surface stress tensor.

When not doing a sweep over several frequencies, or for other reasons, a specific Fourier component of magnetic force can be accessed directly by using the same variable in its standard form; **mmtp.nTX_1_fft0**, **mmtp.nTX_1_fft1**, **mmtp.nTX_1_fft2**, … **mmtp.nTX_1_fftn**, where **n** would be one less half the number of time frames solved for.

ACOUSTICAL CONSIDERATIONS

The acoustical domain in this example is everything outside the motor housing. Perfectly Matched Layer feature is used on outer domains to absorb outgoing wave energy and thus simulating the emission of acoustic noise in an open space without reflections. On boundaries between the Perfectly Matched Layer and the air surrounding the motor housing, an Exterior Field Calculation is applied in order to calculate the sound pressure level at any distance outside the geometry.

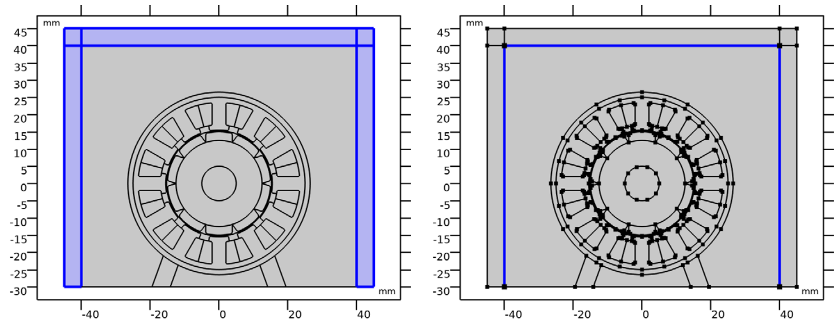


Figure 4: Domains defined by Perfectly Matched Layer and boundaries where Exterior Field Calculation is applied.

The Acoustic-Structure Boundary Multiphysics feature automatically couples the Pressure Acoustics with Solid Mechanics at their common boundaries.

ELECTROMAGNETIC CONSIDERATIONS

Apart from the normal steps of configuring the motor model detailed in step-by-step instructions below, two aspects are highlighted here.

Firstly in order to generate the computation of Maxwell surface stress tensors a Force Calculation feature must be defined on the domains representing the solid for which boundary forces are required.

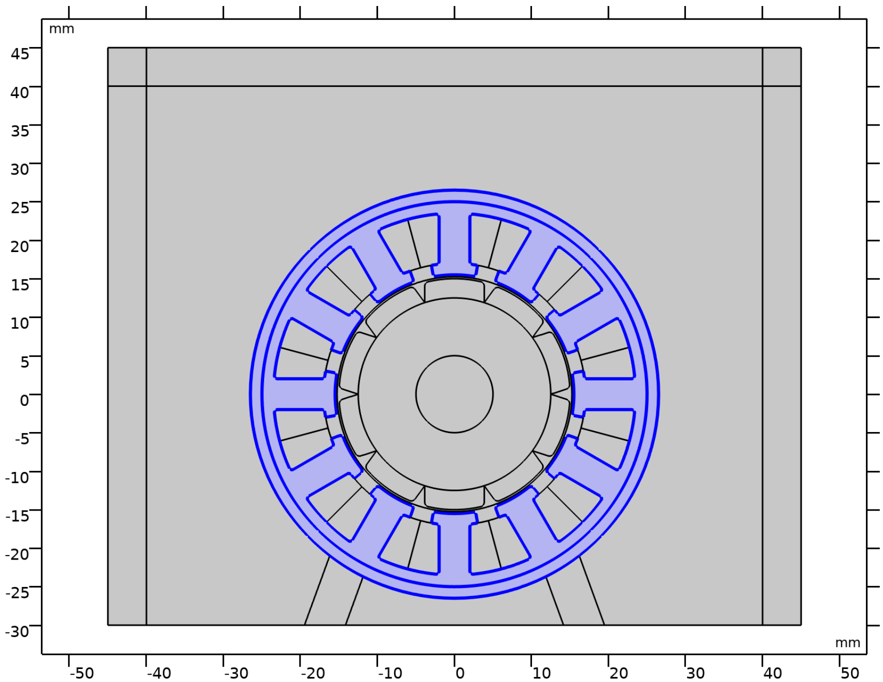


Figure 5: Force Calculation applied to stator core.

The solid components should all be surrounded by air or its equivalent, and not be adjacent to any soft or hard magnetic material. In this case the coils are adjacent stator core but these pose no issue as magnetically they are equivalent to air. A boundary layer mesh is applied to the tips of stator teeth to improve the accuracy of Maxwell surface stress tensor calculation in this area where the magnetic forces are concentrated.

The second matter of extra consideration is the temporal resolution of the electromagnetic simulation. The cogging torque or torque due to reluctance variations as the rotor magnets are passing by stator teeth, will occur on the 12'th harmonic of the fundamental electrical frequency.

$$n_{cog} = \frac{N_s}{gcd(N_s,N_p)} \cdot 2 \tag{1}$$

Here N_s and N_p are the number of slots and poles, and gcd() determines the greatest common divisor of these two integers.

In this example it is assumed for the sake of simplicity and moderate solution time that the impact of induced currents is negligible and the electromagnetic part is solved at the nominal current and shaft speed over one electrical period. Resolving the highest frequency of interest, for example the cogging torque, with six time frames ensures sufficient temporal resolution.

In cases where the impact of induced currents is expected to be significant, it would be necessary to ensure that both excitation and induced currents are periodic over the time period solved for. Furthermore, the electromagnetic problem would need to be solved for at all speeds to be investigated.

Results and Discussion

In Figure 6, the eigenmode of the second natural frequency of the motor structure is shown.

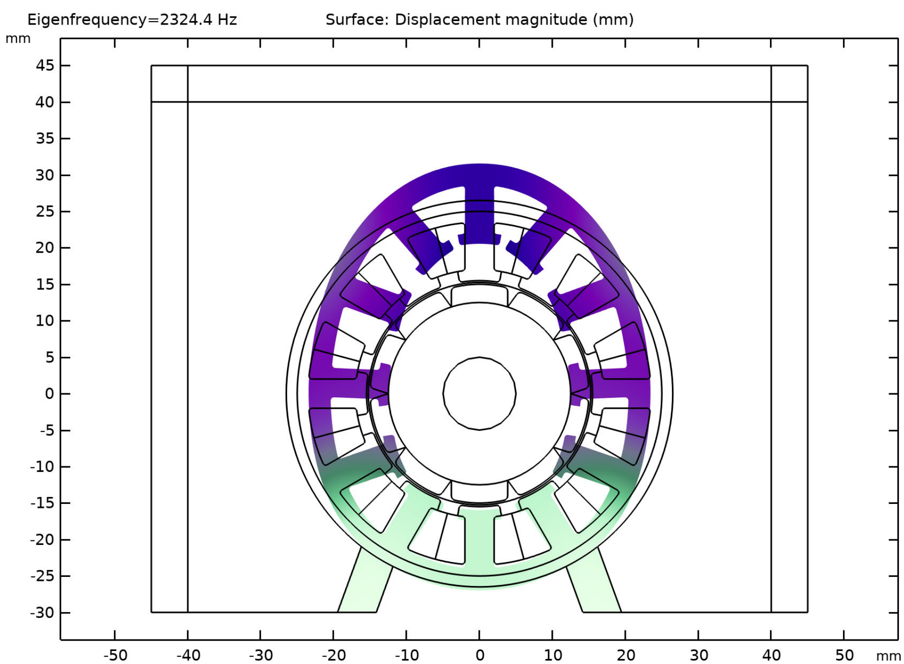


Figure 6: Eigenmode of second natural frequency.

In Figure 7, the 12 first force harmonics are plotted with same scale providing a qualitative comparison. It can be seen the second, fourth and sixth harmonics are the most prominent.

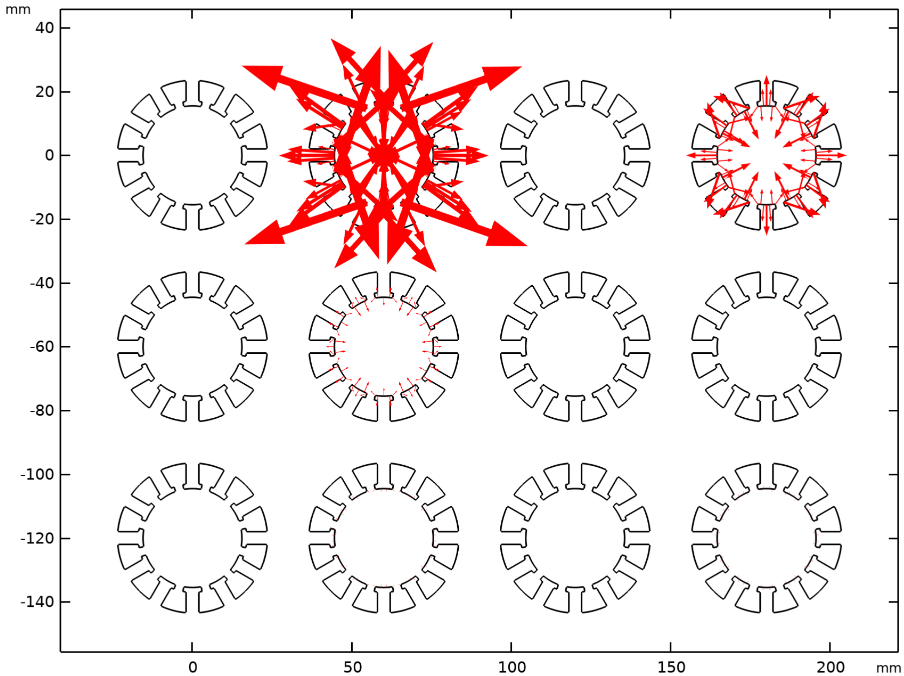


Figure 7: Qualitative evaluation of magnetic force harmonics with the first at top left and 12'th bottom right.

Figure 8 shows the Campbell diagram resulting from the final study. The horizontal lines represent the natural frequencies found in first study. Each of the diagonal lines represent a force harmonic as function of the rotor speed, with the first harmonic to the bottom right. Force harmonics at frequencies below a set limit are not solved for to conserve solution time. The coloring of the diagonal lines represent the sound pressure level measured at point $x = y = 1$ m.

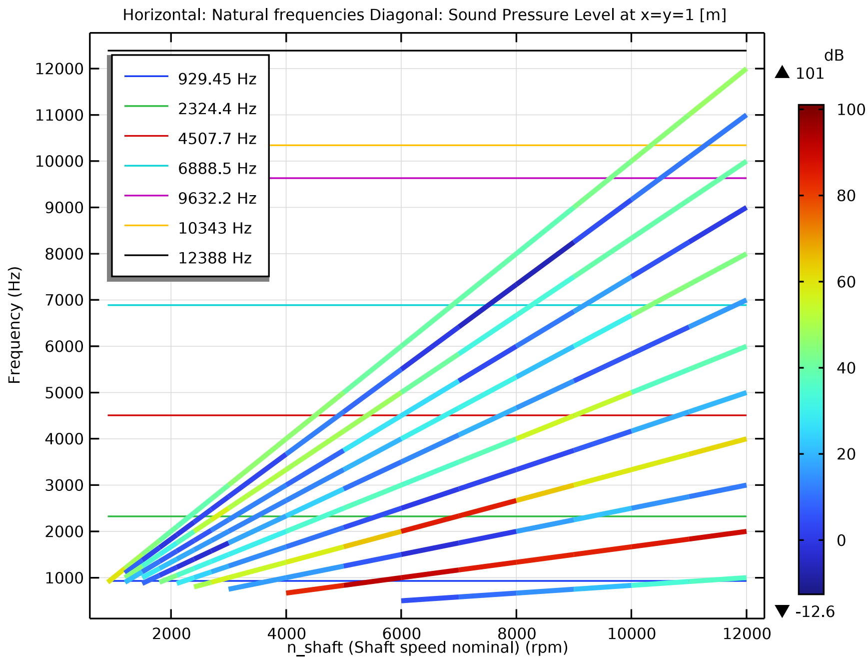


Figure 8: Campbell diagram.

It is clear that the second and fourth are the force harmonics causing the most noise emission. Interestingly the fourth harmonic does not excite the first natural frequency as much as the second, and likewise the sixth harmonic does not excite the second natural frequency as much as the fourth harmonic.

Magnetic forces will have twice the frequency of the magnetic fields that produce them. As such the presence of second and sixth force harmonics are expected being produced by the fundamental electrical frequency and three phase excitation of the stator winding correspondingly. With this understanding the prominence of the fourth harmonic might come as a surprise as it suggests there is a component of the magnetic field varying with twice the fundamental electrical frequency. Although not immediately obvious, one can see this variation on the outer stator core when playing back the animation which is generated at the end of the step-by-step instructions below.

Application Library path: `ACDC_Module/Devices,_Motors_and_Generators/pm_motor_2d_campbell_diagram`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.
2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
5 Click **Add**.
6 In the **Select Physics** tree, select **AC/DC>Electromagnetics and Mechanics> Magnetic Machinery, Rotating, Time Periodic (mmtp)**.
7 Click **Add**.
8 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for **Geometry**, locate the **Units** section.
3 From the **Length unit** list, choose **mm**.
4 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.
5 Browse to the model's Application Libraries folder and double-click the file `pm_motor_2d_campbell_diagram_geom_sequence.mph`.
6 In the **Geometry** toolbar, click **Build All**.
7 In the **Model Builder** window, collapse the **Geometry 1** node.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
`n_shaft`	`12000[rpm]`	200 1/s	`Shaft speed nominal`
`n_cog`	`Ns/gcd(Np,Ns)*2`	12	`First cogging harmonic`
`Nframes`	`n_cog*6`	72	`Number of time frames`
`Ipk`	`1[A]`	1 A	`Phase current`
`init_ang`	`3.45[rad]`	3.45 rad	`Initial current angle for peak torque`
`f_el`	`n_shaft*Np/2`	1000 1/s	`Electrical frequency as function of shaft speed sweep`
`L`	`50[mm]`	0.05 m	`Axial length`

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1, 2, 4, 34, and 35 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 From the **Physics** list, choose **Pressure Acoustics, Frequency Domain (acpr)**.

Disk 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Selections>Disk**.

Next steps will create a few selections handy for configuring physics and mesh later on.

2 In the **Settings** window for **Disk**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `16.3`.

5 In the **Inner radius** text field, type `15.1`.

Adjacent 1

1. In the **Definitions** toolbar, click **Adjacent**.
2. In the **Settings** window for **Adjacent**, locate the **Input Entities** section.
3. Under **Input selections**, click **Add**.
4. In the **Add** dialog box, select **Stator iron (External Stator – Slotted 1)** in the **Input selections** list.
5. Click **OK**.

Disk 2

1. In the **Model Builder** window, right-click **Selections** and choose **Disk**.
2. In the **Settings** window for **Disk**, locate the **Geometric Entity Level** section.
3. From the **Level** list, choose **Boundary**.
4. Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.
5. Under **Selections**, click **Add**.
6. In the **Add** dialog box, select **Adjacent 1** in the **Selections** list.
7. Click **OK**.
8. In the **Settings** window for **Disk**, locate the **Size and Shape** section.
9. In the **Outer radius** text field, type 24.

DEFINITIONS

In the **Model Builder** window, collapse the **Component 1 (comp1)>Definitions** node.

ADD MATERIAL FROM LIBRARY

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

1. Go to the **Add Material** window.
2. In the tree, select **Built-in>Air**.
3. Click **Add to Component** in the window toolbar.
4. In the tree, select **Built-in>Steel AISI 4340**.
5. Click **Add to Component** in the window toolbar.
6. In the tree, select **AC/DC>Soft Iron (Without Losses)**.
7. Click **Add to Component** in the window toolbar.
8. In the tree, select **AC/DC>Hard Magnetic Materials> Sintered NdFeB Grades (Chinese Standard)>N42 (Sintered NdFeB)**.

9 Click **Add to Component** in the window toolbar.

10 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Steel AISI 4340 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Steel AISI 4340 (mat2)**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `10, 27,36` in the **Selection** text field.

5 Click **OK**.

Soft Iron (Without Losses) (mat3)

1 In the **Model Builder** window, click **Soft Iron (Without Losses) (mat3)**.

2 Select Domains 6 and 48 only.

3 In the **Settings** window for **Material**, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	185[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	7500	kg/m³	Basic

N42 (Sintered NdFeB) (mat4)

The electrical conductivity of magnets is zeroed out to exclude any effects of induced currents.

1 In the **Model Builder** window, click **N42 (Sintered NdFeB) (mat4)**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Rotor_magnets (Internal Rotor – Surface Mounted Magnets 1)**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`0*1/1.4[uohm*m]`	S/m	Basic

5 Click the **Colors** button in the **Graphics** toolbar.

6 In the **Graphics** window toolbar, click ▼ next to **Colors**, then choose **Show Material Color and Texture**.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Structural domains**.

4 Locate the **Thickness** section. In the *d* text field, type `L`.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundaries 18 and 66 only.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

2 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Acoustic domains**.

Sound Hard Boundary (Wall) 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Sound Hard Boundary (Wall)**.

2 In the **Settings** window for **Sound Hard Boundary (Wall)**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `2,7,78,80` in the **Selection** text field.

5 Click **OK**.

Exterior Field Calculation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Exterior Field Calculation**.

2 Select Boundaries 6, 9, and 79 only.

MAGNETIC MACHINERY, ROTATING, TIME PERIODIC (MMTP)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Machinery, Rotating, Time Periodic (mmtp)**.

2 In the **Settings** window for **Magnetic Machinery, Rotating, Time Periodic**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Electromagnetic domains**.

4 Locate the **Thickness** section. In the d text field, type `L`.

5 Locate the **Time Periodic Settings** section. In the f_{TP} text field, type `f_el`.

6 In the n_{TP} text field, type `Nframes`.

7 Locate the **Motion Settings** section. In the n_{poles} text field, type `Np`.

Rotational Continuity Pair 1

In the **Physics** toolbar, click **Pairs** and choose **Rotational Continuity Pair**.

Rotating Domain 1

The Rotating Domain feature will by default select all domains on one side of the Rotational Continuity Pair, which in this case are the rotor domains as wanted. Further the default Time periodic rotation will ensure rotation through an angle corresponding to a pole pair or one period of the fundamental electrical frequency.

In the **Physics** toolbar, click **Domains** and choose **Rotating Domain**.

Laminated Core 1

1 In the **Physics** toolbar, click **Domains** and choose **Laminated Core**.

2 Select Domains 6 and 48 only.

Magnet 1

1 In the **Physics** toolbar, click **Domains** and choose **Magnet**.

2 In the **Settings** window for **Magnet**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Rotor_magnets (Internal Rotor – Surface Mounted Magnets 1)**.

4 Locate the **Magnet** section. From the **Pattern type** list, choose **Circular pattern**.

5 From the **Type of periodicity** list, choose **Alternating**.

North 1

1 In the **Model Builder** window, expand the **Magnet 1** node, then click **North 1**.

2 Select Boundary 357 only.

South 1

1 In the **Model Builder** window, click **South 1**.

2 Select Boundary 354 only.

Multiphase Winding 1

1 In the **Physics** toolbar, click **Domains** and choose **Multiphase Winding**.

2 In the **Settings** window for **Multiphase Winding**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Stator slots (External Stator – Slotted 1)**.

4 Locate the **Multiphase Winding** section. In the I_{pk} text field, type `Ipk`.

5 In the α_i text field, type `init_ang`.

The default Winding electrical frequency corresponds with the frequency of the Magnetic Machinery Time Periodic interface which is what is intended in this example. If solving for a longer period however, the Winding electrical frequency can be set explicitly here.

6 From the **Winding layout configuration** list, choose **Automatic three phase**.

7 In the n_{slots} text field, type `Ns`.

8 Click **Add Phases**.

Force Calculation 1

1 In the **Physics** toolbar, click **Domains** and choose **Force Calculation**.

2 Select Domains 6 and 36 only.

ADD MULTIPHYSICS

1 In the **Physics** toolbar, click **Add Multiphysics** to open the **Add Multiphysics** window.

2 Go to the **Add Multiphysics** window.

3 Find the **Select the physics interfaces you want to couple** subsection. In the table, clear the **Couple** check box for **Magnetic Machinery, Rotating, Time Periodic (mmtp)**.

4 In the tree, select **Acoustics>Acoustic–Structure Interaction>Acoustic–Solid Interaction, Frequency Domain**.

5 Click **Add to Component** in the window toolbar.

6 In the **Physics** toolbar, click **Add Multiphysics** to close the **Add Multiphysics** window.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 In the table, clear the **Use** check box for **Pressure Acoustics, Frequency Domain (acpr)**.

4 Right-click **Component 1 (comp1)>Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `2.5`.

5 In the **Curvature factor** text field, type `0.5`.

6 In the **Resolution of narrow regions** text field, type `0.3`.

Size 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.

2 Drag and drop **Size 1** below **Size**.

3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Domain**.

5 Select Domain 6 only.

6 Locate the **Element Size** section. Click the **Custom** button.

7 Locate the **Element Size Parameters** section.

8 Select the **Maximum element size** check box. In the associated text field, type `0.7`.

Free Triangular 1

1 In the **Model Builder** window, click **Free Triangular 1**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `3,5-15,17-33,36-49` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Free Triangular**, click **Build All**.

Boundary Layers 1

1 In the **Mesh** toolbar, click **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 3, 13, and 37 only.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Disk 1**.

4 Locate the **Layers** section. In the **Number of layers** text field, type 1.

5 In the **Thickness adjustment factor** text field, type 5.

Boundary Layer Properties 1

1 In the **Mesh** toolbar, click **More Attributes** and choose **Boundary Layer Properties**.

2 Select Boundaries 6, 9, and 79 only.

3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

4 In the **Number of layers** text field, type 1.

5 In the **Thickness adjustment factor** text field, type 5.

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1, 2, 4, 34, and 35 only.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 4, 8, and 82 only.

3 In the **Settings** window for **Distribution**, click **Build All**.

MESH 1

In the **Model Builder** window, collapse the **Component 1 (comp1)>Mesh 1** node.

ADD STUDY

1 In the **Home** toolbar, click **Windows** and choose **Add Study**.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Eigenfrequency**.

4 Click **Add Study** in the window toolbar.

5 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Stationary**.

6 Click **Add Study** in the window toolbar.

7 In the **Select Study** tree, select **General Studies>Frequency Domain**.

8 Click **Add Study** in the window toolbar.

9 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 1

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 1** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

3 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type 7.

4 Clear the **Search for eigenfrequencies around shift** check box.

5 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Pressure Acoustics, Frequency Domain (acpr)**.

6 In the table, clear the **Solve for** check box for **Acoustic–Structure Boundary 1 (asb1)**.

7 In the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (solid)

In the **Settings** window for **2D Plot Group**, click **Plot Next**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
freq_min	900[Hz]	900 Hz	Treshold frequency for sweep

STUDY 2

Step 1: Stationary

1. In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.
2. In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3. In the table, clear the **Solve for** check box for **Solid Mechanics (solid)**.
4. In the **Home** toolbar, click **Compute**.

SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

Boundary Load 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2. In the **Settings** window for **Boundary Load**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Disk 2**.
4. Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

mmtp.nTX_1_fft(freq/f_el)	x
mmtp.nTY_1_fft(freq/f_el)	y

STUDY 3

After having coupled the magnetic forces as a load in Solid Mechanics it is time to configure the final study to solve for the structural vibrations and acoustical noise for each force harmonic. Generally if the electromagnetic forces are dependent on rotor speed and contrary to this example, the Magnetic Machinery physics would be directly included in the following study (which makes away with Study 2).

Step 1: Frequency Domain

1. In the **Model Builder** window, under **Study 3** click **Step 1: Frequency Domain**.
2. In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.
3. In the table, clear the **Solve for** check box for **Magnetic Machinery, Rotating, Time Periodic (mmtp)**.
4. Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
5. From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 2, Stationary**.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

The range of rotor speeds is configured to have a finer resolution at the lower end where most of the higher force harmonics are passing through the first natural frequencies.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
n_shaft (Shaft speed nominal)	`range(900,300,3000) range(4000,1000,12000)`	`rpm`

5 Click to expand the **Advanced Settings** section. Select the **Reuse solution from previous step** check box.

Step 1: Frequency Domain

Instead of solving for all harmonics at all rotor speeds, it is possible to restrict the frequencies solved for to the ones that reach above a certain level, here `freq_min`. Typically, the lower harmonics do not reach the natural frequencies until higher speeds. If they are deemed uninteresting, they can be excluded from the analysis as follows.

1 In the **Model Builder** window, click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `range(12,-1,floor(max(freq_min/f_el,1)))*f_el`.

4 In the **Study** toolbar, click **Compute**.

RESULTS

Campbell diagram

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Campbell diagram` in the **Label** text field.

Line Segments 1

1 Right-click **Campbell diagram** and choose **Line Segments**.

2 In the **Settings** window for **Line Segments**, locate the **x-Coordinates** section.

3 In the table, enter the following settings:

Expression	Unit	Description
900	1	
12000	1	

4 Click **Add Expression** in the upper-right corner of the **y-Coordinates** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Global>solid.freq - Frequency - Hz**.

5 Click **Add Expression** in the upper-right corner of the **y-Coordinates** section. From the menu, choose **solid.freq - Frequency - Hz**.

6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Horizontal: Natural frequencies`.

8 Click to expand the **Legends** section. Select the **Show legends** check box.

Global 1

1 In the **Model Builder** window, right-click **Campbell diagram** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 3/Parametric Solutions 1 (sol4)**.

4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Global>solid.freq - Frequency - Hz**.

5 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.

6 Click to expand the **Coloring and Style** section. From the **Width** list, choose **3**.

7 Click to expand the **Legends** section. Clear the **Show legends** check box.

Color Expression 1

1 Right-click **Global 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Expression** section.

3 In the **Expression** text field, type `subst(acpr.efc1.Lp_pext,x,1[m],y,1[m])`.

4 From the **Unit** list, choose **dB**.

Campbell diagram

1 In the **Model Builder** window, under **Results** click **Campbell diagram**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Upper left**.

4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

5 Select the **Show units** check box.

6 In the **Campbell diagram** toolbar, click **Plot**.

Global 1

1 In the **Model Builder** window, click **Global 1**.

2 In the **Settings** window for **Global**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Diagonal: Sound Pressure Level at x=y=1 [m]`.

5 In the **Campbell diagram** toolbar, click **Plot**.

Magnetic Flux Density Norm (mmtp)

In the following a few modifications to the default plot of magnetic flux density are made to shorten time taken to generate animation.

Surface 1

1 In the **Model Builder** window, expand the **Magnetic Flux Density Norm (mmtp)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Magnetic Machinery, Rotating, Time Periodic>Magnetic>mmtp.normB_tpph - Magnetic flux density norm, function of phase - T**.

Streamline 1

In the **Model Builder** window, right-click **Streamline 1** and choose **Disable**.

Contour 1

1 In the **Model Builder** window, click **Contour 1**.

2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Magnetic Machinery, Rotating, Time Periodic>Magnetic>mmtp.AZ_tpph - Magnetic vector potential out of plane, function of phase - Wb/m**.

Animation 1

1 In the **Magnetic Flux Density Norm (mmtp)** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Animation Editing** section.

3 From the **Sequence type** list, choose **Dynamic data extension**.

4 Locate the **Frames** section. In the **Number of frames** text field, type `36`.

5 Locate the **Playing** section. From the **Repeat** list, choose **Number of iterations**.

6 In the **Number of iterations** text field, type `5`.

7 Click to expand the **Advanced** section. Clear the **Synchronize scales between frames** check box.

8 Click the **Play** button in the **Graphics** toolbar.

Magnetic force harmonics

The remainder of these instructions is a rather repetitive set of steps in order to reproduce Figure 7 in the Results and Discussion section, and is provided here for reference.

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Magnetic force harmonics` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Line 1

1 Right-click **Magnetic force harmonics** and choose **Line**.

2 In the **Settings** window for **Line**, locate the **Expression** section.

3 In the **Expression** text field, type `1`.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **From theme**.

Selection 1

1 Right-click **Line 1** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Disk 2**.

Arrow Line 1

1 In the **Model Builder** window, right-click **Magnetic force harmonics** and choose **Arrow Line**.

2 In the **Settings** window for **Arrow Line**, locate the **Expression** section.

3 In the **X-component** text field, type `mmtp.nTX_1_fft1`.

4 In the **Y-component** text field, type `mmtp.nTY_1_fft1`.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

6 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `2000`.

Arrow Line 1, Line 1

1 In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 1** and **Arrow Line 1**.

2 Right-click and choose **Duplicate**.

Translation 1

1 In the **Model Builder** window, right-click **Line 2** and choose **Translation**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `60`.

Arrow Line 2

1 In the **Model Builder** window, under **Results>Magnetic force harmonics** click **Arrow Line 2**.

2 In the **Settings** window for **Arrow Line**, locate the **Expression** section.

3 In the **X-component** text field, type `mmtp.nTX_1_fft2`.

4 In the **Y-component** text field, type `mmtp.nTY_1_fft2`.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Arrow Line 1**.

Translation 1

1 Right-click **Arrow Line 2** and choose **Translation**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `60`.

Arrow Line 2, Line 2

1 In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 2** and **Arrow Line 2**.

2 Right-click and choose **Duplicate**.

Arrow Line 3, Line 3

In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 3** and **Arrow Line 3**.

Translation 1

1 In the **Model Builder** window, expand the **Results>Magnetic force harmonics>Line 3** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `120`.

Arrow Line 3

1 In the **Model Builder** window, under **Results>Magnetic force harmonics** click **Arrow Line 3**.

2 In the **Settings** window for **Arrow Line**, locate the **Expression** section.

3 In the **X-component** text field, type `mmtp.nTX_1_fft3`.

4 In the **Y-component** text field, type `mmtp.nTY_1_fft3`.

Translation 1

1 In the **Model Builder** window, expand the **Arrow Line 3** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `120`.

Arrow Line 3, Line 3

1 In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 3** and **Arrow Line 3**.

2 Right-click and choose **Duplicate**.

Arrow Line 4, Line 4

In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 4** and **Arrow Line 4**.

Translation 1

1 In the **Model Builder** window, expand the **Results>Magnetic force harmonics>Line 4** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `180`.

Arrow Line 4

1 In the **Model Builder** window, under **Results>Magnetic force harmonics** click **Arrow Line 4**.

2 In the **Settings** window for **Arrow Line**, locate the **Expression** section.

3 In the **X-component** text field, type `mmtp.nTX_1_fft4`.

4 In the **Y-component** text field, type `mmtp.nTY_1_fft4`.

Translation 1

1 In the **Model Builder** window, expand the **Arrow Line 4** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `180`.

Arrow Line 4, Line 4

1 In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 4** and **Arrow Line 4**.

2 Right-click and choose **Duplicate**.

Arrow Line 5, Line 5

In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 5** and **Arrow Line 5**.

Translation 1

1 In the **Model Builder** window, expand the **Results>Magnetic force harmonics>Line 5** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `0`.

4 In the **y** text field, type `-60`.

Arrow Line 5

1 In the **Model Builder** window, under **Results>Magnetic force harmonics** click **Arrow Line 5**.

2 In the **Settings** window for **Arrow Line**, locate the **Expression** section.

3 In the **X-component** text field, type `mmtp.nTX_1_fft5`.

4 In the **Y-component** text field, type `mmtp.nTY_1_fft5`.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Translation 1

1 In the **Model Builder** window, expand the **Arrow Line 5** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `0`.

4 In the **y** text field, type `-60`.

Arrow Line 5

Click the **Zoom Extents** button in the **Graphics** toolbar.

Arrow Line 5, Line 5

1 In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 5** and **Arrow Line 5**.

2 Right-click and choose **Duplicate**.

Arrow Line 6, Line 6

In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 6** and **Arrow Line 6**.

Translation 1

1 In the **Model Builder** window, expand the **Results>Magnetic force harmonics>Line 6** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `60`.

Arrow Line 6

1 In the **Model Builder** window, under **Results>Magnetic force harmonics** click **Arrow Line 6**.

2 In the **Settings** window for **Arrow Line**, locate the **Expression** section.

3 In the **X-component** text field, type `mmtp.nTX_1_fft6`.

4 In the **Y-component** text field, type `mmtp.nTY_1_fft6`.

Translation 1

1 In the **Model Builder** window, expand the **Arrow Line 6** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `60`.

Arrow Line 6, Line 6

1 In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 6** and **Arrow Line 6**.

2 Right-click and choose **Duplicate**.

Arrow Line 7, Line 7

In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 7** and **Arrow Line 7**.

Translation 1

1 In the **Model Builder** window, expand the **Results>Magnetic force harmonics>Line 7** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `120`.

Arrow Line 7

1 In the **Model Builder** window, under **Results>Magnetic force harmonics** click **Arrow Line 7**.

2 In the **Settings** window for **Arrow Line**, locate the **Expression** section.

3 In the **X-component** text field, type `mmtp.nTX_1_fft7`.

4 In the **Y-component** text field, type `mmtp.nTY_1_fft7`.

5 Click the  **Zoom Extents** button in the **Graphics** toolbar.

Translation 1

1 In the **Model Builder** window, expand the **Arrow Line 7** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type 120.

Arrow Line 7, Line 7

1 In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 7** and **Arrow Line 7**.

2 Right-click and choose **Duplicate**.

Arrow Line 8, Line 8

In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 8** and **Arrow Line 8**.

Translation 1

1 In the **Model Builder** window, expand the **Results>Magnetic force harmonics>Line 8** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type 180.

Arrow Line 8

1 In the **Model Builder** window, under **Results>Magnetic force harmonics** click **Arrow Line 8**.

2 In the **Settings** window for **Arrow Line**, locate the **Expression** section.

3 In the **X-component** text field, type `mmtp.nTX_1_fft8`.

4 In the **Y-component** text field, type `mmtp.nTY_1_fft8`.

Translation 1

1 In the **Model Builder** window, expand the **Arrow Line 8** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type 180.

Arrow Line 8, Line 8

1 In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 8** and **Arrow Line 8**.

2 Right-click and choose **Duplicate**.

Arrow Line 9, Line 9

In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 9** and **Arrow Line 9**.

Translation 1

1 In the **Model Builder** window, expand the **Results>Magnetic force harmonics>Line 9** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `0`.

4 In the **y** text field, type `-120`.

Arrow Line 9

1 In the **Model Builder** window, under **Results>Magnetic force harmonics** click **Arrow Line 9**.

2 In the **Settings** window for **Arrow Line**, locate the **Expression** section.

3 In the **X-component** text field, type `mmtp.nTX_1_fft9`.

4 In the **Y-component** text field, type `mmtp.nTY_1_fft9`.

Translation 1

1 In the **Model Builder** window, expand the **Arrow Line 9** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `0`.

4 In the **y** text field, type `-120`.

Arrow Line 9, Line 9

1 In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 9** and **Arrow Line 9**.

2 Right-click and choose **Duplicate**.

Arrow Line 10, Line 10

In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 10** and **Arrow Line 10**.

Translation 1

1 In the **Model Builder** window, expand the **Results>Magnetic force harmonics>Line 10** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `60`.

Arrow Line 10

1 In the **Model Builder** window, under **Results>Magnetic force harmonics** click **Arrow Line 10**.

2 In the **Settings** window for **Arrow Line**, locate the **Expression** section.

3 In the **X-component** text field, type `mmtp.nTX_1_fft10`.

4 In the **Y-component** text field, type `mmtp.nTY_1_fft10`.

Translation 1

1 In the **Model Builder** window, expand the **Arrow Line 10** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `60`.

4 Click the **Zoom Extents** button in the **Graphics** toolbar.

Arrow Line 10, Line 10

1 In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 10** and **Arrow Line 10**.

2 Right-click and choose **Duplicate**.

Arrow Line 11, Line 11

In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 11** and **Arrow Line 11**.

Translation 1

1 In the **Model Builder** window, expand the **Results>Magnetic force harmonics>Line 11** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `120`.

Arrow Line 11

1 In the **Model Builder** window, under **Results>Magnetic force harmonics** click **Arrow Line 11**.

2 In the **Settings** window for **Arrow Line**, locate the **Expression** section.

3 In the **X-component** text field, type `mmtp.nTX_1_fft11`.

4 In the **Y-component** text field, type `mmtp.nTY_1_fft11`.

Translation 1

1 In the **Model Builder** window, expand the **Arrow Line 11** node, then click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `120`.

Arrow Line 11, Line 11

1 In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 11** and **Arrow Line 11**.

2 Right-click and choose **Duplicate**.

Arrow Line 12, Line 12

In the **Model Builder** window, under **Results>Magnetic force harmonics**, Ctrl-click to select **Line 12** and **Arrow Line 12**.

Line 12

In the **Model Builder** window, expand the **Results>Magnetic force harmonics>Line 12** node.

Translation 1

1 In the **Model Builder** window, expand the **Results>Magnetic force harmonics>Arrow Line 12** node, then click **Results>Magnetic force harmonics>Line 12>Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `180`.

Arrow Line 12

1 In the **Model Builder** window, under **Results>Magnetic force harmonics** click **Arrow Line 12**.

2 In the **Settings** window for **Arrow Line**, locate the **Expression** section.

3 In the **X-component** text field, type `mmtp.nTX_1_fft12`.

4 In the **Y-component** text field, type `mmtp.nTY_1_fft12`.

Translation 1

1 In the **Model Builder** window, click **Translation 1**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `180`.

Arrow Line 1

1 In the **Model Builder** window, under **Results>Magnetic force harmonics** click **Arrow Line 1**.

2 In the **Settings** window for **Arrow Line**, locate the **Coloring and Style** section.

3 Select the **Scale factor** check box. In the associated text field, type `1e-4`.

4 In the **Magnetic force harmonics** toolbar, click **Plot**.

RESULTS

Magnetic force harmonics

In the **Model Builder** window, collapse the **Results>Magnetic force harmonics** node.

Campbell diagram

1 In the **Model Builder** window, click **Campbell diagram**.

2 In the **Campbell diagram** toolbar, click **Plot**.