Created in COMSOL Multiphysics 6.0

# Linear Motor in 2D

## *Introduction*

This example shows how to build and analyze a model of a linear motor, a device exerting force in order to produce translational motion. Typical characteristics of linear motors are high precision and quick acceleration. They are used in a wide variety of applications ranging from small actuators to propulsion of transportation systems such as Maglev trains. Here we will consider a flat and slotted topology with a three phase excitation in the stationary part, often called *forcer*. The other part often called *track* will be the moving part containing permanent magnets which makes out for a brush-less and synchronous design. A 3D representation of the linear motor is illustrated in Figure 1.

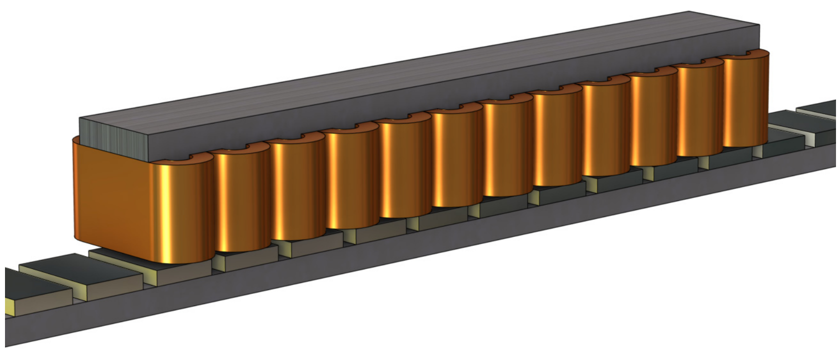

*Figure 1: A 3D representation of the magnetic parts of the linear motor to be simulated.*

A vertical cut along the axis of motion gives a cross section where the currents are perpendicular to the plane and where the major magnetic field gradients vary in-plane, which is in line with 2D magnetic field formulation.

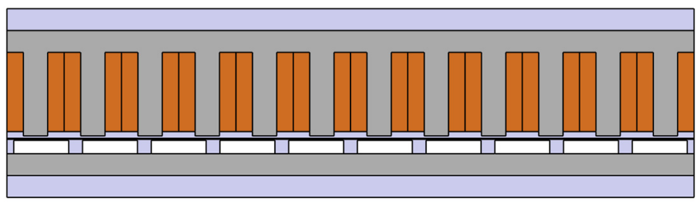

*Figure 2: 2D geometry of linear motor with magnetic steel materials in grey, coils in orange and permanent magnets in white.*

In principle, a cross section of a linear machine is very similar to a 2D representation of an axial flux machine. A circular surface parallel to the axis of rotation (and in line with the magnetic flux direction), will capture the magnetic field gradients in plane and have perpendicular currents. This linear motor modeling example doubles as a 2D representation of an axial flux machine.

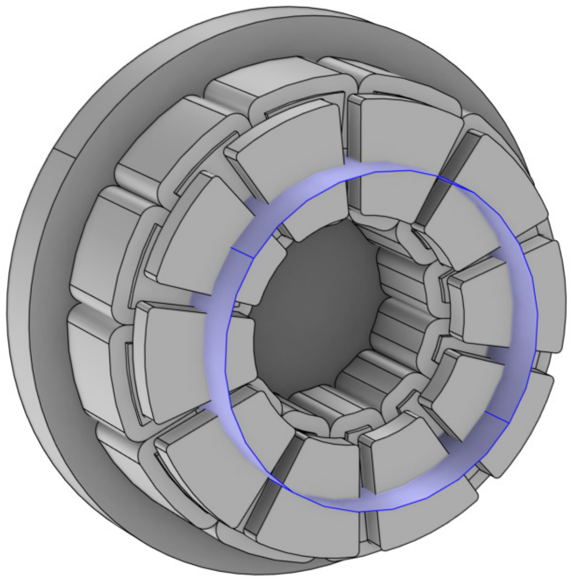

*Figure 3: The blue circular surface rolled out can represent a cross section of an axial flux motor in 2D.*

For the combined model of a linear and an axial flux motor, consider a fractional slot topology. This means that the number of slots divided by the number of phases and the number of poles is less than one, and that coils do not span around more than one tooth. In terms of a linear motor, the geometry represents a periodic section of the machine. In terms of an axial flux motor, the geometry can represent the entire circular cross section.

For simplicity, consider a low but reasonable number of 12 slots and 10 poles on a periodic and symmetric geometry spanning approximately 150 mm in the direction of linear motion. For an axial flux motor, the modeled surface is preferably centered radially in the magnetic region, which in this case represents a surface diameter of roughly 50 mm.

The simulation comprises two studies, where the first one determines the optimal current angle, and the second one simulates linear motion in time. The resulting air-gap shear stress, total force and torque, energy balance, and harmonics are post-processed and analyzed.

## *Modeling in COMSOL Multiphysics*

In order to properly account for the interaction between the stationary and moving parts, the geometry is split into two objects called an *assembly* in COMSOL Multiphysics. In this example, the split is created along a horizontal line centered in the middle of the air gap. The Moving Mesh interface lets you define the motion of the moving part.

The parts are coupled on their adjacent boundaries in the middle of the air gap by the Periodic Pair feature. The Periodic Pair feature performs a transformation of the field crossing the boundaries accounting for the instantaneous velocity. Where the boundaries are not adjacent to each other, the Periodic Pair maps the field back together assuming symmetric geometry. For the external vertical boundaries on each side, a Periodic Condition feature is applied for both stationary and moving parts.

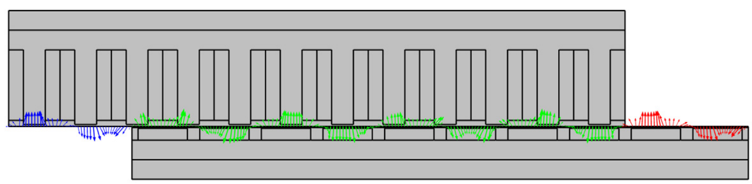

*Figure 4: Arrows showing magnetic flux density and direction on boundary in middle of air gap. Field protruding stationary part (red arrows) are mapped onto stationary part (blue arrows).*

In the air gap, a thin domain is added in order to accurately compute the linear force produced. An integral of the magnetic force density in the direction of motion over this domain provides a robust way of calculating the shear force acting on the parts.

The optimal current angle is determined by a stationary simulation keeping the moving part at the initial position, while sweeping the current vector over one electrical period. The initial current angle constant is then updated with an angle corresponding to the maximum force. The motion of the moving part is defined such that it moves in synchronization and in the same direction as the winding magnetic field.

In this example, the motion is prescribed as a constant velocity, while it is also possible to add equations governing a dynamic motion — a result of forces, friction and mass of inertia.

## *Results and Discussion*

Figure 5 shows a plot of the magnetic flux density giving an impression of the field distribution at a given instant.

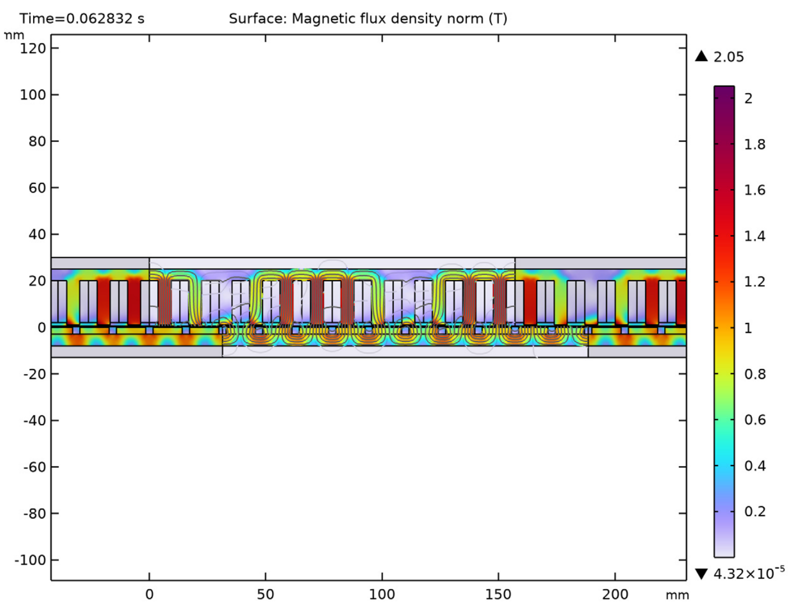


*Figure 5: Magnetic flux density distribution at the end of first electrical period (darker regions on each side are post-processed displacement of modeled result).*

[Figure 6](#) shows the linear force as a function of time for one electrical period. The linear force is on average 56 N and has the 12th harmonic of electrical frequency as the most prominent ripple.

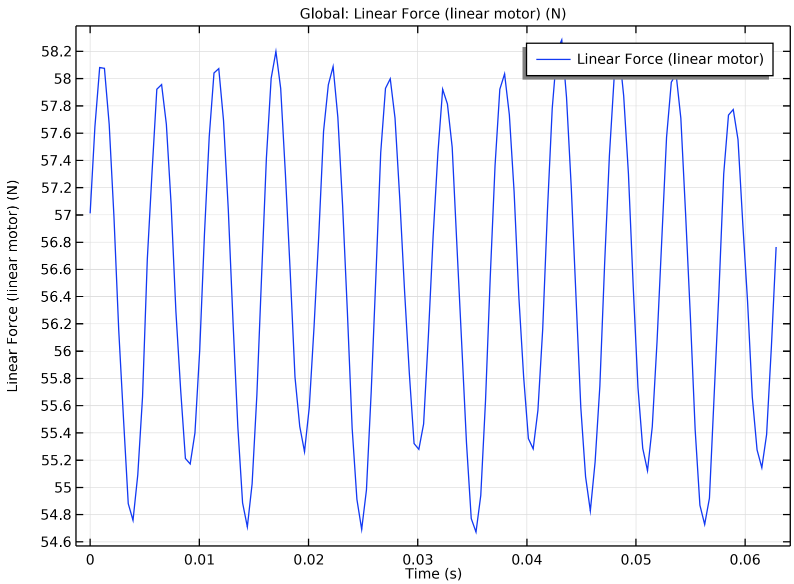


*Figure 6: Linear force as function of time for one electrical period.*

A Fourier transform of the linear force signal reveals that the 12th harmonic is about eight times larger than the 6th harmonic, as shown in Figure 7.

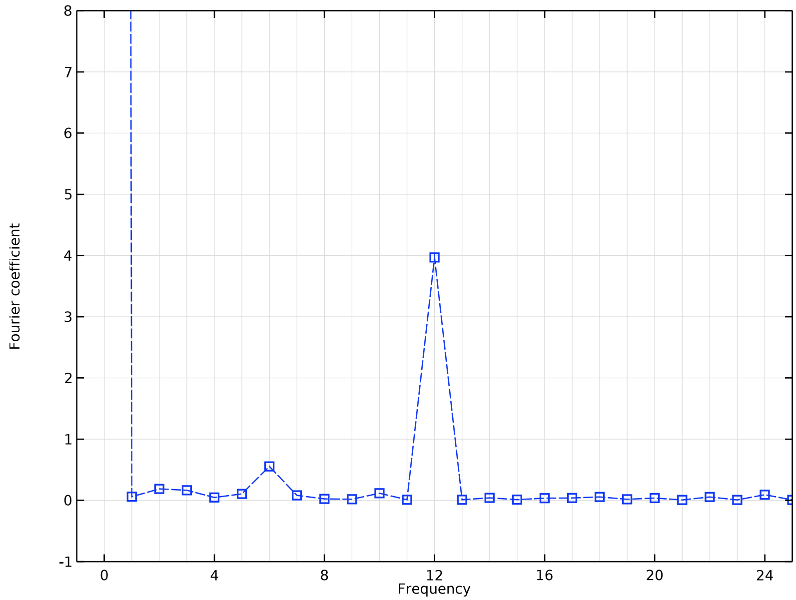


*Figure 7: Harmonic content of the linear force with the electrical frequency as basis.*

The overall results are evaluated in an **Evaluation Group** which allows for a tabulated presentation of quantities of different units.

TABLE 1: RESULTS SUMMARY.

| QUANTITY | VALUE |
|---|---|
| Airgap shear stress (Pa) | 17990 |
| Linear force (N) | 56.5 |
| Torque (axial flux motor) (N·m) | 1.4 |
| Linear motion power (W) | 28.3 |
| Rotational motion power (W) | 28.3 |
| Electrical input power (W) | 38.0 |
| Windings resistive loss (W) | 9.7 |
| Magnets induced loss (W) | 0.05 |

**Application Library path:** ACDC_Module/Motors_and_Actuators/linear_motor_2d

## *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 **AC/DC>Electromagnetic Fields>Magnetic Fields (mf)**.

3 Click **Add**.

4 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**.

### GLOBAL DEFINITIONS

#### *Parameters - main*

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

2 In the **Settings** window for **Parameters**, type Parameters - main in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

| Name | Expression | Value | Description |
|---|---|---|---|
| Np | 10 | 10 | Nr of poles on moving part/ rotor |
| Ns | 12 | 12 | Nr of stator slots |
| v_lin | 0.5[m/s] | 0.5 m/s | Linear motion velocity |
| I_ph | 2.5[A] | 2.5 A | Phase current rms |

| Name | Expression | Value | Description |
|---|---|---|---|
| Nturn | 120 | 120 | Nr of turns per coil |
| r_inc | 25[mm] | 0.025 m | Incision/ computational plane radius (axial flux motor) |
| s_rot | v_lin/(2*pi* r_inc) | 3.1831 1/s | Rotational speed (axial flux motor) |
| f_el | s_rot*Np/2 | 15.915 1/s | Electrical frequency |
| ang_el_init | 0[deg] | 0 rad | Initial current angle (max force/ torque) |
| L_mag | 20[mm] | 0.02 m | Length into plane/ radial extent of magnetic materials |
| t | 0[s] | 0 s | Time variable alias |

*Parameters - geometry*

1 In the **Home** toolbar, click Pi **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type Parameters - geometry in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

| Name | Expression | Value | Description |
|---|---|---|---|
| arc_inc | r_inc*2*pi[rad] | 0.15708 m | Incision plane arc length |
| th_air | 5[mm] | 0.005 m | Air thickness outside |
| th_rbyoke | 5[mm] | 0.005 m | Rotor back-yoke thickness |
| th_mag | 3[mm] | 0.003 m | Magnet thickness |
| th_airgap | 1[mm] | 0.001 m | Airgap thickness |
| th_stooth | 19[mm] | 0.019 m | Stator toooth thickness |
| th_coil | 18[mm] | 0.018 m | Coil thickness |
| th_sbyoke | 5[mm] | 0.005 m | Stator back-yoke thickness |
| ang_mag | 2*pi[rad]/Np*0.8 | 0.50265 rad | Magnet pitch angle |

| Name | Expression | Value | Description |
|---|---|---|---|
| arc_mag | r_inc*ang_mag | 0.012566 m | Magnet arc length |
| arc_pp | arc_inc/Np | 0.015708 m | Pole pitch arc length |
| w_slot | 7.5[mm] | 0.0075 m | Slot width |
| ang_slot | 2*asin(w_slot/2/r_inc) | 0.30114 rad | Slot pitch angle |
| arc_slot | r_inc*ang_slot | 0.0075284 m | Slot arc length |
| arc_stooth | arc_inc/Ns-arc_slot | 0.0055616 m | Stator tooth arc length |
| A_wire | th_coil*w_slot/2/Nturn*0.55 | 3.0938E-7 m² | Wire cross-sectional area including slot fill factor of .55 |

*Variables 1*

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

| Name | Expression | Unit | Description |
|---|---|---|---|
| iA | I_ph*cos(2*pi*f_el*t+ang_el_init) | A | A phase current |
| iB | I_ph*cos(2*pi*f_el*t+ang_el_init-2*pi/3) | A | B phase current |
| iC | I_ph*cos(2*pi*f_el*t+ang_el_init-4*pi/3) | A | C phase current |

The geometry construction is colored by it being a representation of a 3D axial flux motor. This means that the width and height of a rectangle is treated as ’arc length’ and ’thickness’ correspondingly.

## GEOMETRY 1

*Rectangle 1 (r1)*

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type arc_mag.

4 In the **Height** text field, type th_mag.

5 Locate the **Position** section. In the **x** text field, type (arc_pp-arc_mag)/2.

**6** In the **y** text field, type -th_mag.

*Move 1 (mov1)*

**1** In the **Geometry** toolbar, click **Transforms** and choose **Move**.

**2** Select the object **r1** only.

**3** In the **Settings** window for **Move**, locate the **Displacement** section.

**4** In the **x** text field, type arc_pp.

**5** Locate the **Input** section. Select the **Keep input objects** check box.

*Magnets up*

**1** In the **Geometry** toolbar, click **Transforms** and choose **Array**.

**2** In the **Settings** window for **Array**, type Magnets up in the **Label** text field.

**3** Select the object **r1** only.

**4** Locate the **Size** section. In the **x size** text field, type Np/2.

**5** Locate the **Displacement** section. In the **x** text field, type arc_pp*2.

**6** Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

*Magnets down*

**1** In the **Geometry** toolbar, click **Transforms** and choose **Array**.

**2** In the **Settings** window for **Array**, type Magnets down in the **Label** text field.

**3** Select the object **mov1** only.

**4** Locate the **Size** section. In the **x size** text field, type Np/2.

**5** Locate the **Displacement** section. In the **x** text field, type arc_pp*2.

**6** Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

*All magnets*

**1** In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

**2** In the **Settings** window for **Union Selection**, type All magnets in the **Label** text field.

**3** Locate the **Input Entities** section. Click **Add**.

**4** In the **Add** dialog box, in the **Selections to add** list, choose **Magnets up** and **Magnets down**.

**5** Click **OK**.

*Rotor back yoke*

**1** In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, type `Rotor back yoke` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `arc_inc`.

4 In the **Height** text field, type `th_rbyoke`.

5 Locate the **Position** section. In the **y** text field, type `-th_rbyoke-th_mag`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

*Rotor air*

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, type `Rotor air` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `arc_inc`.

4 In the **Height** text field, type `th_mag+th_rbyoke+th_air+th_airgap/2`.

5 Locate the **Position** section. In the **y** text field, type `-th_mag-th_rbyoke-th_air`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

*Aux torque calc domain*

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, type `Aux torque calc domain` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `arc_inc`.

4 In the **Height** text field, type `th_airgap/4`.

5 Locate the **Position** section. In the **y** text field, type `th_airgap/4`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

*Rotor selection*

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, type `Rotor selection` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Object**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Magnets up**, **Magnets down**, **Rotor back yoke**, **Rotor air**, and **Aux torque calc domain**.

6 Click **OK**.

*Rotor*

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 In the **Settings** window for **Union**, type `Rotor` in the **Label** text field.

3 Locate the **Union** section. From the **Input objects** list, choose **Rotor selection**.

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

*Stator tooth*

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, type `Stator tooth` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `arc_stooth`.

4 In the **Height** text field, type `th_stooth`.

5 Locate the **Position** section. In the **x** text field, type `arc_slot/2`.

6 In the **y** text field, type `th_airgap`.

7 Click **Build Selected**.

*Stator teeth*

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 In the **Settings** window for **Array**, type `Stator teeth` in the **Label** text field.

3 Select the object **r5** only.

4 Locate the **Size** section. In the **x size** text field, type `Ns`.

5 Locate the **Displacement** section. In the **x** text field, type `arc_stooth+arc_slot`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

7 From the **Show in physics** list, choose **Off**.

*Stator back yoke*

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, type `Stator back yoke` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `arc_inc`.

4 In the **Height** text field, type `th_sbyoke`.

5 Locate the **Position** section. In the **y** text field, type `th_airgap+th_stooth`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

7 From the **Show in physics** list, choose **Off**.

*Stator yoke selection*

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, type `Stator yoke selection` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Object**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Stator teeth** and **Stator back yoke**.

6 Click **OK**.

*Stator yoke*

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 In the **Settings** window for **Union**, type `Stator yoke` in the **Label** text field.

3 Locate the **Union** section. From the **Input objects** list, choose **Stator yoke selection**.

4 Clear the **Keep interior boundaries** check box.

*Coil leg*

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, type `Coil leg` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `arc_slot/2`.

4 In the **Height** text field, type `th_coil`.

5 Locate the **Position** section. In the **y** text field, type `th_airgap+th_stooth-th_coil`.

*Coil legs left*

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 In the **Settings** window for **Array**, type `Coil legs left` in the **Label** text field.

3 Select the object **r7** only.

4 Locate the **Size** section. In the **x size** text field, type `Ns`.

5 Locate the **Displacement** section. In the **x** text field, type `arc_stooth+arc_slot`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

7 From the **Show in physics** list, choose **Off**.

*Coil legs right*

1 In the **Geometry** toolbar, click **Transforms** and choose **Move**.

2 In the **Settings** window for **Move**, type `Coil legs right` in the **Label** text field.

3 Locate the **Input** section. From the **Input objects** list, choose **Coil legs left**.
4 Select the **Keep input objects** check box.
5 Locate the **Displacement** section. In the **x** text field, type `arc_slot/2+arc_stooth`.
6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.
7 From the **Show in physics** list, choose **Off**.

*Coil domains*

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.
2 In the **Settings** window for **Union Selection**, type `Coil domains` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Object**.
4 Locate the **Input Entities** section. Click **Add**.
5 In the **Add** dialog box, in the **Selections to add** list, choose **Coil legs left** and **Coil legs right**.
6 Click **OK**.
7 In the **Settings** window for **Union Selection**, locate the **Resulting Selection** section.
8 From the **Show in physics** list, choose **Domain selection**.

*Stator air*

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, type `Stator air` in the **Label** text field.
3 Locate the **Size and Shape** section. In the **Width** text field, type `arc_inc`.
4 In the **Height** text field, type `th_airgap/2+th_stooth+th_sbyoke+th_air`.
5 Locate the **Position** section. In the **y** text field, type `th_airgap/2`.
6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.
7 From the **Show in physics** list, choose **Off**.

*Stator selection*

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.
2 In the **Settings** window for **Union Selection**, type `Stator selection` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Object**.
4 Locate the **Input Entities** section. Click **Add**.
5 In the **Add** dialog box, in the **Selections to add** list, choose **Stator yoke selection**, **Coil legs left**, **Coil legs right**, and **Stator air**.

6 Click **OK**.

*Stator*

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 In the **Settings** window for **Union**, type `Stator` in the **Label** text field.

3 Locate the **Union** section. From the **Input objects** list, choose **Stator selection**.

*Form Union (fin)*

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 In the **Geometry** toolbar, click **Build All**.

## GEOMETRY 1

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

## DEFINITIONS

*Magnetic Steel domains*

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Magnetic Steel domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **Rotor back yoke** and **Stator yoke selection**.

5 Click **OK**.

*Airgap integration*

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

Next we will create an integration operating on the thin domain of the air gap which will assist in accurate force/torque calculation.

2 In the **Settings** window for **Integration**, type `Airgap integration` in the **Label** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **Aux torque calc domain**.

*Variables 2*

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

| Name | Expression | Unit | Description |
|---|---|---|---|
| F_phi | mf.Bx*mf.By/mu0_const | Pa | Shear stress density |
| Torque | intop1(F_phi*r_inc)/(th_airgap/4)*L_mag | J | Torque (axial flux motor) |
| Force | intop1(F_phi)/(th_airgap/4)*L_mag | N | Linear Force (linear motor) |

*Torque*

1 In the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

Two probes are added to calculate force and torque while solving.

2 In the **Settings** window for **Global Variable Probe**, type `Torque` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `Torque`.

4 Click to expand the **Table and Window Settings** section.

*Shear Force*

1 In the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type `Shear Force` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `Force`.

4 Locate the **Table and Window Settings** section. Click **Add Plot Window**.

## DEFINITIONS

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

## MATERIALS

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 **AC/DC>Soft Iron (Without Losses)**.

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

6 In the tree, select **AC/DC>Hard Magnetic Materials> Sintered NdFeB Grades (Chinese Standard)>N40M (Sintered NdFeB)**.

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

8 In the tree, select **AC/DC>Copper**.

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

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

## MATERIALS

### *Soft Iron (Without Losses) (mat2)*

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Soft Iron (Without Losses) (mat2)**.

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

3 From the **Selection** list, choose **Magnetic Steel domains**.

### *N40M (Sintered NdFeB) (mat3)*

1 In the **Model Builder** window, click **N40M (Sintered NdFeB) (mat3)**.

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

3 From the **Selection** list, choose **All magnets**.

### *Copper (mat4)*

1 In the **Model Builder** window, click **Copper (mat4)**.

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

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

## COMPONENT 1 (COMP1)

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

2 In the **Settings** window for **Component**, in the **Graphics** window toolbar, click ▼ next to **Colors**, then choose **Show Material Color and Texture**.

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

4 In the **Definitions** toolbar, click **Moving Mesh** and choose **Domains>Prescribed Deformation**.

## MOVING MESH

### *Moving part/ rotor*

1 In the **Settings** window for **Prescribed Deformation**, type `Moving part/ rotor` in the **Label** text field.

2 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Rotor selection**.

**3** Locate the **Prescribed Deformation** section. Specify the *dx* vector as

| | |
|---|---|
| v_lin*t | X |
| 0 | Y |

**4** In the **Model Builder** window, collapse the **Moving Mesh** node.

## MATERIALS

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

## MAGNETIC FIELDS (MF)

**1** In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields (mf)**.

**2** In the **Settings** window for **Magnetic Fields**, locate the **Thickness** section.

**3** In the *d* text field, type `L_mag`.

In this model, the **Out-of-plane thickness:** will affect the resistance and voltage in coil features.

**4** Click to expand the **Discretization** section. From the **Magnetic vector potential** list, choose **Linear**.

### *Periodic Pair 1*

**1** In the **Physics** toolbar, click **Pairs** and choose **Periodic Pair**.

Next we will add the **Periodic Pair** feature and configure it to operate on the **Identity Boundary Pair** which was created as final step in geometry sequence.

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

**3** Under **Pairs**, click **Add**.

**4** In the **Add** dialog box, select **Identity Boundary Pair 1 (ap1)** in the **Pairs** list.

**5** Click **OK**.

### *Periodic Condition - stator*

**1** In the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.

**2** In the **Settings** window for **Periodic Condition**, type `Periodic Condition - stator` in the **Label** text field.

**3** Select Boundaries 64, 66, 68, 70, and 190–193 only.

### *Periodic Condition - moving part/ rotor*

**1** In the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.

**2** In the **Settings** window for **Periodic Condition**, type `Periodic Condition - moving part/ rotor` in the **Label** text field.

3 Select Boundaries 1, 3, 5, 7, and 60–63 only.

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

*Magnetic steel*

1 In the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 In the **Settings** window for **Ampère's Law**, type `Magnetic steel` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Magnetic Steel domains**.

4 Locate the **Constitutive Relation B-H** section. From the **Magnetization model** list, choose **B-H curve**.

5 Locate the **Material Type** section. From the **Material type** list, choose **Solid**.

*A phase winding*

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

2 In the **Settings** window for **Coil**, type `A phase winding` in the **Label** text field.

3 Select Domains 16, 19, 28–31, 40, and 41 only.

4 Locate the **Coil** section. From the **Conductor model** list, choose **Homogenized multiturn**.

5 Select the **Coil group** check box.

6 In the $I_{coil}$ text field, type `iA`.

7 Locate the **Homogenized Multiturn Conductor** section. In the $N$ text field, type `Nturn`.

8 In the $a_{coil}$ text field, type `A_wire`.

*Reversed Current Direction 1*

1 In the **Physics** toolbar, click **Attributes** and choose **Reversed Current Direction**.

2 Select Domains 19, 29, 30, and 40 only.

*B phase winding*

1 In the **Model Builder** window, right-click **A phase winding** and choose **Duplicate**.

2 In the **Settings** window for **Coil**, type `B phase winding` in the **Label** text field.

3 Locate the **Domain Selection** section. Click **Clear Selection**.

4 Select Domains 20–23 and 32–35 only.

5 Locate the **Coil** section. In the $I_{coil}$ text field, type `iB`.

*Reversed Current Direction 1*

1 In the **Model Builder** window, expand the **B phase winding** node, then click **Reversed Current Direction 1**.

2 In the **Settings** window for **Reversed Current Direction**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domains 21, 22, 32, and 35 only.

*C phase winding*

1 In the **Model Builder** window, right-click **B phase winding** and choose **Duplicate**.

2 In the **Settings** window for **Coil**, type `C phase winding` in the **Label** text field.

3 Locate the **Domain Selection** section. Click **Clear Selection**.

4 Select Domains 24–27 and 36–39 only.

5 Locate the **Coil** section. In the $I_{\mathrm{coil}}$ text field, type `iC`.

*Reversed Current Direction 1*

1 In the **Model Builder** window, expand the **C phase winding** node, then click **Reversed Current Direction 1**.

2 In the **Settings** window for **Reversed Current Direction**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domains 24, 27, 37, and 38 only.

In order to ensure zero net current flowing through magnets, two **Coil** features, one for each magnet direction, with a zero current specified for the **Coil current** setting are added.

*Magnets up*

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

2 In the **Settings** window for **Coil**, type `Magnets up` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Magnets up**.

4 Locate the **Material Type** section. From the **Material type** list, choose **Solid**.

5 Locate the **Coil** section. In the $I_{\mathrm{coil}}$ text field, type `0[A]`.

6 Locate the **Constitutive Relation B-H** section. From the **Magnetization model** list, choose **Remanent flux density**.

7 Specify the **e** vector as

| | |
|---|---|
| 0 | X |
| 1 | Y |
| 0 | Z |

*Magnets down*

**1** Right-click **Magnets up** and choose **Duplicate**.

**2** In the **Settings** window for **Coil**, type `Magnets down` in the **Label** text field.

**3** Locate the **Domain Selection** section. From the **Selection** list, choose **Magnets down**.

**4** Locate the **Constitutive Relation B-H** section. Specify the **e** vector as

| | |
|---|---|
| 0 | X |
| -1 | Y |
| 0 | Z |

**5** In the **Model Builder** window, collapse the **Magnetic Fields (mf)** node.

The default mesh is slightly refined in order to properly resolve the force harmonics appearing in the air gap.

### MESH 1

*Size*

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Edit Physics-Induced Sequence**.

*Size 1 - periodic boundaries*

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

**2** In the **Settings** window for **Size**, type `Size 1 - periodic boundaries` in the **Label** text field.

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

**4** Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

**5** In the associated text field, type `2[mm]`.

*Size 1 - magnetic steel*

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

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

**3** In the **Settings** window for **Size**, type `Size 1 - magnetic steel` in the **Label** text field.

**4** Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.

**5** From the **Selection** list, choose **Magnetic Steel domains**.

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

**7** Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

8 In the associated text field, type `2[mm]`.

*Size 2 - airgap*

1 Right-click **Mesh 1** and choose **Size**.

2 In the **Settings** window for **Size**, type `Size 2 - airgap` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 15 only.

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

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated text field, type `0.5[mm]`.

8 Click **Build All**.

### MESH 1

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

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

### ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

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

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

### STUDY 1 - CURRENT ANGLE SWEEP

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

2 In the **Settings** window for **Study**, type `Study 1 - Current angle sweep` in the **Label** text field.

*Step 1: Stationary*

1 In the **Model Builder** window, under **Study 1 - Current angle sweep** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

**5** In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| ang_el_init (Initial current angle (max force/ torque)) | range(0,10,360) | deg |

### STUDY 1 - CURRENT ANGLE SWEEP

**1** In the **Model Builder** window, collapse the **Study 1 - Current angle sweep** node.

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

### RESULTS

*Magnetic Flux Density Norm (mf)*

While solving or after it is completed, click the **Probe Plot 1** tab in the graphics window. Note the value of **ang_el_init** giving the maximum force and update the parameter value.

### GLOBAL DEFINITIONS

*Parameters - main*

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

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

**3** In the table, enter the following settings:

| Name | Expression | Value | Description |
|---|---|---|---|
| ang_el_init | 90[deg] | 1.5708 rad | Initial current angle (max force/ torque) |

### ROOT

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

### ADD STUDY

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

**2** Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

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

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

### STUDY 2 - TRANSIENT

**1** In the **Model Builder** window, click **Study 2**.

**2** In the **Settings** window for **Study**, type `Study 2 - Transient` in the **Label** text field.

*Time Dependent*

1 In the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

For the transient simulation we will modify the default solver settings in order to properly resolve the harmonics we want to investigate. In the following we will let the solver know we need 144 time steps per electrical period, which corresponds to 12 steps for the 12th harmonic of the electrical frequency.

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

3 In the **Output times** text field, type `range(0,1/f_el/144,1/f_el)`.

*Solution 2 (sol2)*

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2 - Transient>Solver Configurations>Solution 2 (sol2)>Time-Dependent Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

5 From the **Jacobian update** list, choose **On every iteration**.

## STUDY 2 - TRANSIENT

1 In the **Model Builder** window, collapse the **Study 2 - Transient** node.

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

## RESULTS

*Magnetic Flux Density Norm (mf) 1*

In the following we will make two duplicates of surface plot and displace them to each side in order to reproduce the figure of magnetic flux density in the results section.

*Surface 2*

1 In the **Model Builder** window, expand the **Magnetic Flux Density Norm (mf) 1** node.

2 Right-click **Results>Magnetic Flux Density Norm (mf) 1>Surface 1** and choose **Duplicate**.

3 In the **Settings** window for **Surface**, click to expand the **Title** section.

4 From the **Title type** list, choose **None**.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **PrismDark**.

6 Clear the **Color legend** check box.

*Translation 1*

1 Right-click **Surface 2** and choose **Translation**.

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

3 In the **x** text field, type `-arc_inc`.

*Surface 3*

In the **Model Builder** window, under **Results>Magnetic Flux Density Norm (mf) 1** right-click **Surface 2** and choose **Duplicate**.

*Translation 1*

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

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

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

4 In the **Magnetic Flux Density Norm (mf) 1** toolbar, click **Plot**.

*Magnetic Flux Density Norm (mf) 1*

Next we will create an **Evaluation Group** so as to gather main results.

*Results summary*

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Results summary` in the **Label** text field.

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

4 Locate the **Transformation** section. Select the **Transpose** check box.

*Surface Average - airgap*

1 Right-click **Results summary** and choose **Average>Surface Average**.

2 In the **Settings** window for **Surface Average**, type `Surface Average - airgap` in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **Aux torque calc domain**.

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

| Expression | Unit | Description |
|---|---|---|
| F_phi | Pa | Airgap shear stress |
| F_phi*arc_inc*L_mag | N | Linear force |
| F_phi*arc_inc*L_mag*r_inc | N*m | Torque (axial flux motor) |

| Expression | Unit | Description |
|---|---|---|
| F_phi*arc_inc*L_mag*v_lin | W | Linear motion power |
| F_phi*arc_inc*L_mag*r_inc*s_rot*2*pi[rad] | W | Rotational motion power |

5 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Average**.

*Global Evaluation - windings*

1 In the **Model Builder** window, right-click **Results summary** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Global Evaluation - windings` in the **Label** text field.

Since all coil features differ only by phase offset we can simply multiply the result from a single coil with three when calculating average values.

3 Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| mf.PCoil_1*3 | W | Electrical input power |
| mf.ICoil_1^2*mf.RCoil_1*3 | W | Windings resistive loss |

4 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Average**.

*Surface Integration - magnets*

1 Right-click **Results summary** and choose **Integration>Surface Integration**.

2 In the **Settings** window for **Surface Integration**, type `Surface Integration - magnets` in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **All magnets**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Magnetic Fields>Heating and losses>mf.Qrh - Volumetric loss density, electric - W/m³**.

5 Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| mf.Qrh*L_mag | W | Magnets induced loss |

6 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Average**.

7 In the **Results summary** toolbar, click **Evaluate**.

*Linear force*

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

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

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

*Global 1*

**1** Right-click **Linear force** and choose **Global**.

**2** In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions> Variables>Force - Linear Force (linear motor) - N**.

**3** In the **Linear force** toolbar, click **Plot**.

*Torque harmonics*

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

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

**3** Locate the **Data** section. From the **Dataset** list, choose **None**.

*Table Graph 1*

**1** Right-click **Torque harmonics** and choose **Table Graph**.

When performing the Fourier transform we will scale the x-axis in order to get the results as harominces of the electrical frequency.

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

**3** From the **x-axis data** list, choose **Time (s)**.

**4** From the **Plot columns** list, choose **Manual**.

**5** In the **Columns** list, select **Torque (axial flux motor) (J)**.

**6** From the **Transformation** list, choose **Discrete Fourier transform**.

**7** From the **Show** list, choose **Frequency spectrum**.

**8** Click to expand the **Preprocessing** section. Find the **x-axis column** subsection. From the **Preprocessing** list, choose **Linear**.

**9** In the **Scaling** text field, type `f_el`.

**10** Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

**11** Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

**12** From the **Positioning** list, choose **In data points**.

**13** In the **Torque harmonics** toolbar, click **Plot**.

*Torque harmonics*

1 In the **Model Builder** window, click **Torque harmonics**.

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

3 Select the **Manual axis limits** check box.

4 In the **x minimum** text field, type -1.

5 In the **x maximum** text field, type 25.

6 In the **y minimum** text field, type -1.

7 In the **y maximum** text field, type 8.

8 Locate the **Grid** section. Select the **Manual spacing** check box.

9 In the **Torque harmonics** toolbar, click **Plot**.