Created in COMSOL Multiphysics 6.2

Permanent Magnet Motor with Efficiency Map

Introduction

This example demonstrates how to model the interaction between electromagnetic losses and temperature in an electrical motor. Firstly the convergence of electromagnetic losses with temporal resolution is investigated. Secondly a sweep over a range of speeds, torque levels and temperatures is performed in order to create the motor efficiency map. The focus of this tutorial is on the multiphysics coupling and so the electromagnetic and thermal aspects not directly related to this are simplified.

Model Definition

The electromagnetic part of this example is modeled using the Magnetic Machinery Time Periodic interface which directly solves for the steady-state operation. This means it solves for the amount of time in which the electromagnetic field is periodic or repeating itself without resolving any startup transients. With a time periodic solution the space dependent time average electromagnetic losses are easily included in the Heat Transfer interface with the Electromagnetic Heating multiphysics coupling. In return the temperature is used to govern the electrical resistance of stator winding and the remanent flux density of rotor magnets, both of which will impact the motor efficiency.

ELECTROMAGNETIC CONSIDERATIONS

The motor geometry is reduced to the smallest sector which can represent the spatial periodicity of the magnetic field as shown in Figure 1. For most rotating machinery designs the number of sectors corresponding with field periodicity is generally found by $N_{\mathrm{sec}} = \mathrm{gcd}(N_p, N_s)$, where N_p and N_s are the number of poles, and where $\mathrm{gcd}()$ finds the greatest common divisor of these integers.

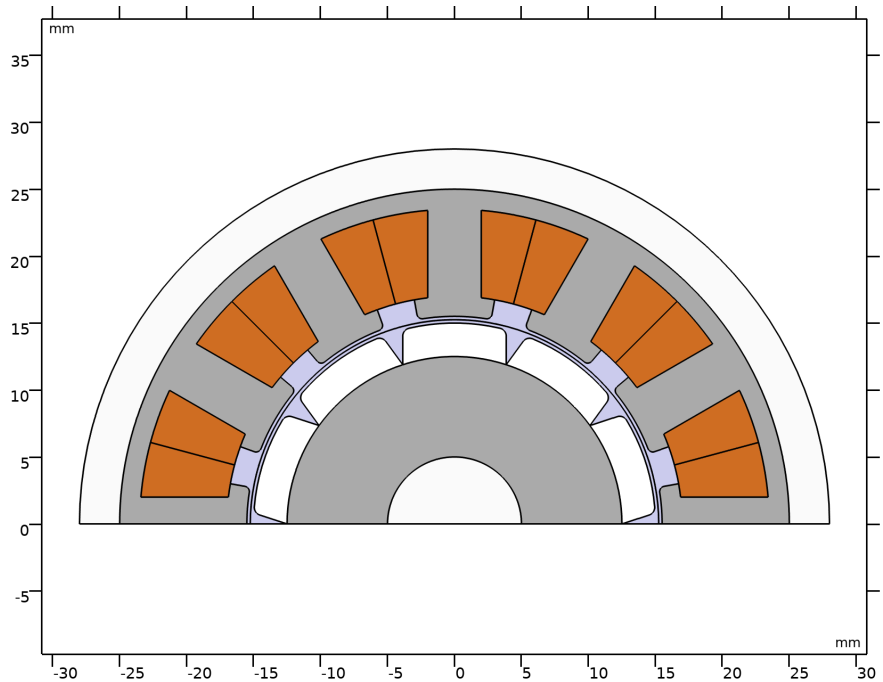

Figure 1: Modeled motor geometry capturing spatial periodicity of the magnetic field.

The time period for which the magnetic field is repeating itself is for most synchronous machine cases equivalent to the period of electrical excitation. In a synchronous machine the excitation frequency is given by $f_{\mathrm{el}} = \omega_{\mathrm{rot}} \cdot N_p/2$ where ω_{rot} is the shaft speed, and hence the excitation time period is $\tau_{\mathrm{el}} = 1/\omega_{\mathrm{el}}$. This is the time it takes the rotor to rotate an angle spanning exactly one pole pair or two poles.

It is not always however the induced currents have the same time periodicity as the excitation even for a synchronous machine. Isolated the induced currents tend to be periodic with the time it takes the rotor to rotate through an angle of $360°/N_{\mathrm{sec}}$, or corresponding with the geometrical periodicity.

This means the periodic time of the induced currents in this model is $\tau_{\mathrm{ind}} = 1/(\omega_{\mathrm{rot}} \cdot N_{\mathrm{sec}})$ $= 2.5/f_{\mathrm{el}}$, which unfortunately does not encompass an integer number of excitation periods. The time period at which both the induced currents and the excitation are periodic will in this case correspond to the time taken for a full mechanical revolution $\tau_{\mathrm{all}} = 1/\omega_{\mathrm{rot}}$. Solving for a full mechanical revolution can be quite resource demanding in terms of solution time and memory requirements however. Hence a study of the temporal resolution or number of time frames required to achieve good convergence of key results is performed.

The coils are represented as homogenized multiturn conductors which is a fair simplification when the conductor cross section is far smaller than the conductor skin depth. This means all coil conductors are evenly distributed inside a single domain representing the coil cross section. When defining the temperature dependent electrical resistivity for such a coil the average temperature of the same domain should be used:

$$\rho_{\mathrm{coil}}(T) = \rho_{\mathrm{ref}} \cdot (1 + \alpha \cdot (\mathtt{aveop1}(T) - T_{\mathrm{ref}})) . \tag{1}$$

The temperature dependence of magnet remanent flux density can be expressed similarly:

$$B_r(T) = B_{\mathrm{r\_ref}} \cdot (1 + \alpha_{\mathrm{Br}} \cdot (T - T_{\mathrm{ref}})) . \tag{2}$$

Here, $B_{\mathrm{r\_ref}}$ and T_{ref} represent the reference flux density and temperature, and α_{Br} represents the remanent flux reversible temperature coefficient of the magnet material.

THERMAL CONSIDERATIONS

The Electromagnetic Heating multiphysics coupling automatically configures the space dependent electromagnetic loss as a volumetric heat source in the Heat Transfer in Solids interface. This can also be done manually by adding a Heat Source feature and specifying the Electromagnetic Volumetric loss density variable **mmtp.Qh** as a General source for the relevant domains.

In an electrical motor there are several thermal barriers which contribute significantly to temperature distribution but which are very thin compared to other geometrical details. Typical examples are the insulation around coils and the thermal contact between stator core and motor housing. One way to account for this without resorting to a very detailed mesh is to specify these thermal barriers with the Thin Layer feature.

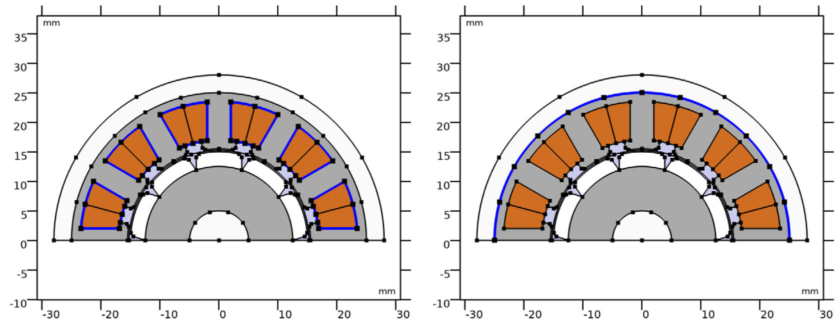

Figure 2: Thin layer accounts for the thermal barrier around coils and between stator core and motor housing.

The cooling of the motor is in this example simplified to Heat Flux boundary conditions on the outer surface of the motor housing and on internal surfaces adjacent to the airgap between rotor and stator. For the airgap cooling the average temperature of stator solid materials is used as External temperature, or temperature of air in airgap, and a modest heat transfer coefficient of 50 $W/(m^2 \cdot K)$ is used to represent moderate cooling by forced convection of air on these boundaries.

Results and Discussion

When varying the number of time frames solved for in the Magnetic Machinery Time Periodic interface, it is clear that a coarse temporal resolution overestimates the torque and consequently the shaft power, and underestimates the losses in most components. On the other hand, while solutions with finer temporal resolution converges both in terms of output power and losses, they will require substantial solution time to compute an entire efficiency map. To strike a balance between accuracy and computational cost using 120 time frames seems appropriate in this particular case. The necessary number of frames might be different for other motor designs and is particularly influenced by the time periodicity of the quantities of interest.

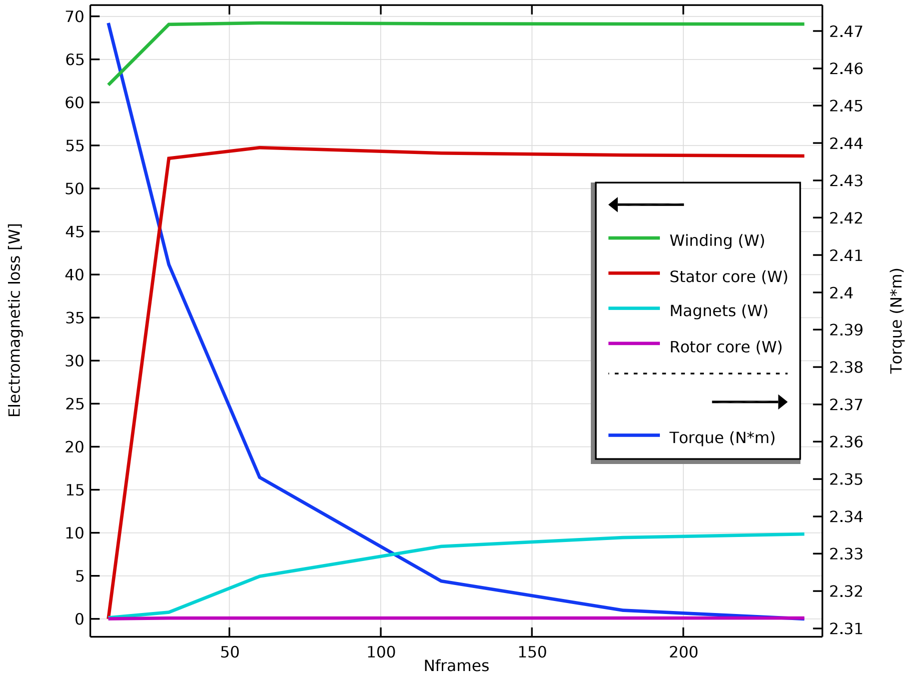

Figure 3: Torque and losses as function of number of time frames for one revolution.

In order to generate an efficiency map, the electromagnetic and thermal simulation is run for a range of speeds and current amplitudes spanning the operational space intended for the motor. A plot of electromagnetic loss distribution of the four corner operating points of this space provides an impression of the magnitude of the key loss mechanisms at play.

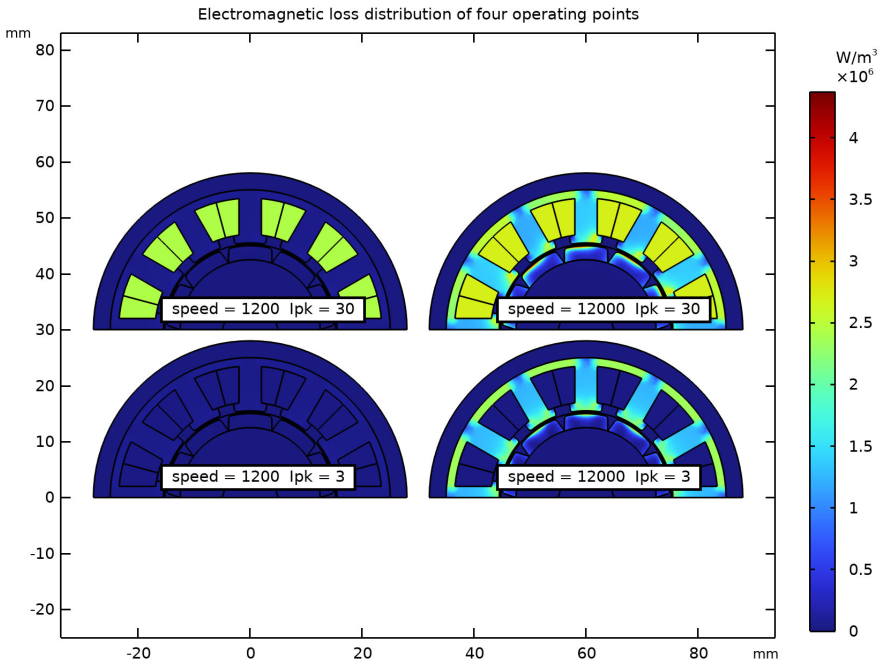

Figure 4: Electromagnetic loss distribution of four operating points.

It can be seen that the *current driven* losses are primarily contained in the winding in top left plot of Figure 4. On the other hand the speed driven losses are prominent in both magnets and stator core in bottom right plot of Figure 4. When these two loss mechanisms are combined in top right plot of Figure 4, it is interesting to note a slight asymmetric distribution of losses in magnets and at tip of stator teeth coinciding with the anticlockwise rotational direction.

A similar plot of the temperature distribution provides insight into the effect these losses has on the temperature of the different components.

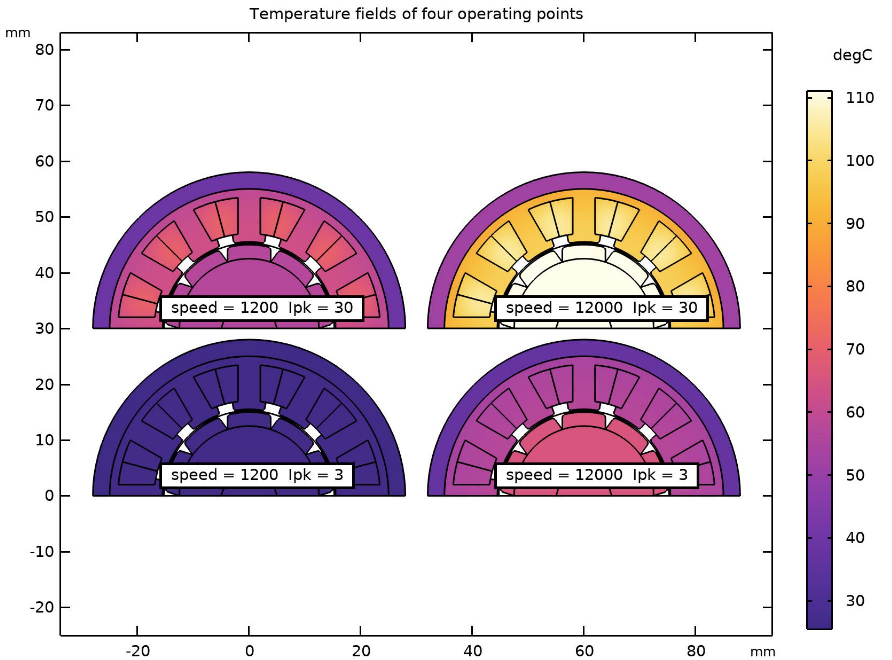

Figure 5: Temperature distribution of four operating points.

As seen in Figure 5, the temperature of rotor and most importantly magnets is dependent on the stator temperature as it is only cooled by air in the airgap.

The efficiency map is generated by solving for four different speeds and five different current levels giving a total of 20 operating points.

.

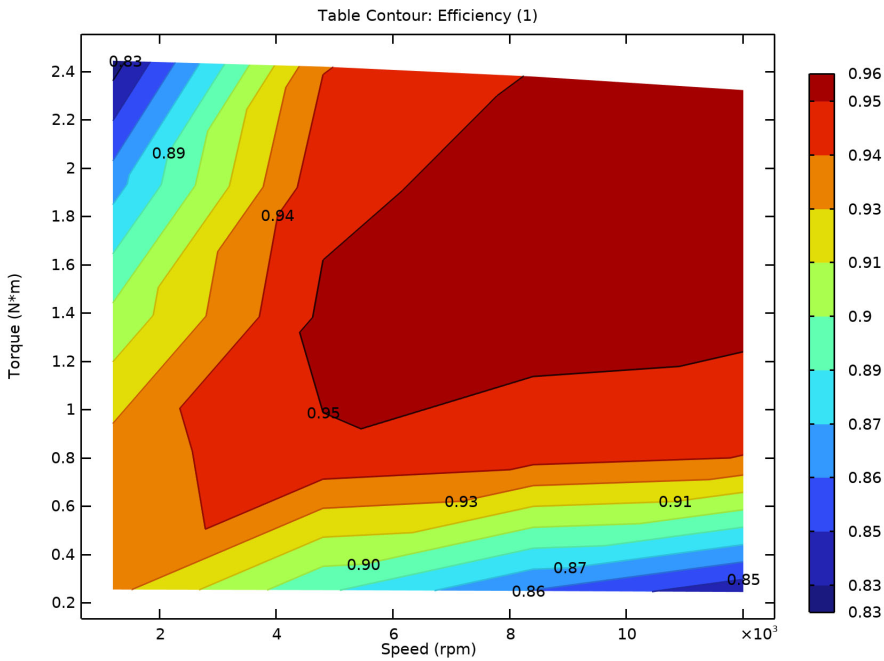

Figure 6: Electromagnetic loss distribution of four operating points.

The efficiency in Figure 6, is plotted against torque and speed and shows a slanted top boundary. This shows the effect of diminishing motor torque for the same amount of stator current as the temperature of the magnets elevate with increasing rotor speed.

Application Library path: `ACDC_Module/Devices,_Motors_and_Generators/pm_motor_2d_efficiency_map`

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 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 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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `pm_motor_2d_efficiency_map_parameters.txt`.

PART LIBRARIES

1 In the **Home** toolbar, click **Windows** and choose **Part Libraries**.

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

3 In the **Part Libraries** window, select **AC/DC Module>Rotating Machinery 2D>Rotors> Internal>surface_mounted_magnet_internal_rotor_2d** in the tree.

4 Click **Add to Geometry**.

GEOMETRY 1

Internal Rotor – Surface Mounted Magnets 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Internal Rotor – Surface Mounted Magnets 1 (pi1)**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
number_of_poles	Np	10	Number of magnetic poles in rotor
number_of_modeled_poles	Np/Nsec	5	Number of magnetic poles included in the geometry

4 Click to expand the **Domain Selections** section. In the table, select the **Keep** check boxes for **Shaft**, **Rotor iron**, **Rotor_magnets**, **Rotor solid domains**, **Rotor air**, and **All**.

PART LIBRARIES

1 In the **Home** toolbar, click **Part Libraries**.

2 In the **Part Libraries** window, select **AC/DC Module>Rotating Machinery 2D>Stators>External>slotted_external_stator_2d** in the tree.

3 Click **Add to Geometry**.

GEOMETRY 1

External Stator – Slotted 1 (pi2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **External Stator – Slotted 1 (pi2)**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
number_of_slots	Ns	12	Number of slots in stator
number_of_modeled_slots	Ns/Nsec	6	Number of slots included in the geometry
slot_winding_type	2	2	Slot winding type: 1-No partition, 2-Radial partition, 3-Azimuthal partition, 4-Radial and azimuthal partition.

4 Locate the **Domain Selections** section. In the table, select the **Keep** check boxes for **Stator iron** and **Stator slots**.

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 **Home** toolbar, click **Build All**.

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

Next, create a few selections and operators that will simplify the configuration of the physics.

DEFINITIONS

Stator Housing

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

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

3 Select Domain 1 only.

Solid Materials

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

2 In the **Settings** window for **Union**, type `Solid Materials` 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 **Stator Housing**, **Shaft (Internal Rotor – Surface Mounted Magnets 1)**, **Rotor iron (Internal Rotor – Surface Mounted Magnets 1)**, **Rotor_magnets (Internal Rotor – Surface Mounted Magnets 1)**, **Stator iron (External Stator – Slotted 1)**, and **Stator slots (External Stator – Slotted 1)**.

5 Click **OK**.

Solid Materials - External Boundaries

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

2 In the **Settings** window for **Adjacent**, type `Solid Materials - External Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Solid Materials** in the **Input selections** list.

5 Click **OK**.

Airgap Heat Flux Boundaries

1 In the **Model Builder** window, right-click **Selections** and choose **Disk**.

2 In the **Settings** window for **Disk**, type `Airgap Heat Flux Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. 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 **Solid Materials - External Boundaries** 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 `18`.

10 In the **Inner radius** text field, type `11`.

11 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside disk**.

Laminated Core - Housing Boundaries

1 Right-click **Selections** and choose **Disk**.

2 In the **Settings** window for **Disk**, type `Laminated Core - Housing Boundaries` in the **Label** text field.

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

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `25.5`.

5 In the **Inner radius** text field, type `24.5`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside disk**.

Winding Insulation Boundaries

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

2 In the **Settings** window for **Adjacent**, type `Winding Insulation Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Stator slots (External Stator – Slotted 1)** in the **Input selections** list.

5 Click **OK**.

Water Jacket - External Boundaries

1 Right-click **Selections** and choose **Disk**.

2 In the **Settings** window for **Disk**, type `Water Jacket - External Boundaries` in the **Label** text field.

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

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `28.8`.

5 In the **Inner radius** text field, type `27.5`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside disk**.

Stator Solid Materials

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

2 In the **Settings** window for **Union**, type `Stator Solid Materials` 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 **Stator Housing**, **Stator iron (External Stator – Slotted 1)**, and **Stator slots (External Stator – Slotted 1)**.

5 Click **OK**.

Average 1 - Winding

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

2 In the **Settings** window for **Average**, type `Average 1 - Winding` in the **Label** text field.

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

Average 2 - Stator Solid Materials

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

2 In the **Settings** window for **Average**, type `Average 2 - Stator Solid Materials` in the **Label** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **Stator Solid Materials**.

ADD PHYSICS

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

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

3 In the tree, select **AC/DC>Electromagnetics and Mechanics>Magnetic Machinery, Rotating, Time Periodic (mmtp)**.

4 Click **Add to Component 1** in the window toolbar.

5 In the tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.

6 Click **Add to Component 1** in the window toolbar.

7 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

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

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

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

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

8 In the tree, select **AC/DC>Hard Magnetic Materials> Sintered NdFeB Grades (Chinese Standard)>N54 (Sintered NdFeB)**.

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

10 In the tree, select **Built-in>High-strength alloy steel**.

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

12 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 Select Domains 2 and 18 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
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	20	W/(m·K)	Basic
Density	rho	1	kg/m³	Basic
Heat capacity at constant pressure	Cp	1	J/(kg·K)	Basic

For the coil domains, specify a value for the thermal conductivity representing the in-plane bulk property of insulated copper strands.

Copper (mat3)

1 In the **Model Builder** window, click **Copper (mat3)**.

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

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

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

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`2[W/(m*K)]`	W/(m·K)	Basic

N54 (Sintered NdFeB) (mat4)

1 In the **Model Builder** window, click **N54 (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
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	9	W/(m·K)	Basic
Density	rho	1	kg/m³	Basic
Heat capacity at constant pressure	Cp	1	J/(kg·K)	Basic

High-strength alloy steel (mat5)

1 In the **Model Builder** window, click **High-strength alloy steel (mat5)**.

2 Select Domains 1 and 20 only.

MATERIALS

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

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

3 In the **Settings** window for **Materials**, in the **Graphics** window toolbar, click ▾ next to **Colors**, then choose **Show Material Color and Texture**.

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

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 **Thickness** section.

3 In the d text field, type `L`.

4 Locate the **Time Periodic Settings** section. In the n_{TP} text field, type `Nframes`.

5 In the f_{TP} text field, type `w_rot`.

6 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**.

Rotational Periodicity 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Rotational Periodicity**.

2 Select Boundaries 1, 2, 10, 30, 31, 36, 121, 123, 127, 129, 132, and 136 only.

Rotating Domain 1

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

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

3 From the **Time periodic rotation** list, choose **Full mechanical revolution**.

Laminated Core 1

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

2 Select Domains 2 and 18 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**.

6 Locate the **Constitutive Relation B-H** section. From the $\|\mathbf{B}_\mathrm{r}\|$ list, choose **User defined**. In the associated text field, type `PM_Br_ref*(1+PM_alpha*(T-PM_Tref))`.

North 1

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

2 Select Boundaries 163, 165, and 166 only.

South 1

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

2 Select Boundary 161 only.

MAGNETIC MACHINERY, ROTATING, TIME PERIODIC (MMTP)

Magnet 1

In the **Model Builder** window, collapse the **Component 1 (comp1)>Magnetic Machinery, Rotating, Time Periodic (mmtp)>Magnet 1** node.

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`.

6 In the f_t text field, type `f_el`.

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

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

9 Click **Add Phases**.

Magnetic Machinery, Rotating, Time Periodic (mmtp)

1 In the **Model Builder** window, collapse the **Component 1 (comp1)>Magnetic Machinery, Rotating, Time Periodic (mmtp)>Multiphase Winding 1** node.

2 In the **Model Builder** window, click **Multiphase Winding 1**.

3 In the **Settings** window for **Multiphase Winding**, locate the **Homogenized Multiturn Conductor** section.

4 In the N text field, type `Nturn`.

5 In the σ_{wire} text field, type `1/(Cu_rho0*(1+Cu_alpha*(aveop1(T)-Cu_Tref)))`.

6 From the **Coil wire cross-section area** list, choose **Filling factor**.

7 In the f text field, type `ff_slot`.

Heat Transfer in Solids (ht)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

2 In the **Settings** window for **Heat Transfer in Solids**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Solid Materials**.

4 Locate the **Physical Model** section. In the d_z text field, type `L`.

5 Click to expand the **Discretization** section. From the **Temperature** list, choose **Linear**.

Thin Layer 1 - Laminated Core <> Housing

1 In the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 In the **Settings** window for **Thin Layer**, type `Thin Layer 1 - Laminated Core <> Housing` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Laminated Core - Housing Boundaries**.

4 Locate the **Shell Properties** section. From the **Shell type** list, choose **Nonlayered shell**. In the L_{th} text field, type `0.5e-4[m]`.

5 Locate the **Heat Conduction** section. From the k list, choose **User defined**. In the associated text field, type `0.02`.

Thin Layer 2 - Winding Insulation

1 In the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 In the **Settings** window for **Thin Layer**, type `Thin Layer 2 - Winding Insulation` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Winding Insulation Boundaries**.

4 Locate the **Shell Properties** section. From the **Shell type** list, choose **Nonlayered shell**. In the L_{th} text field, type `2e-4[m]`.

5 Locate the **Heat Conduction** section. From the k list, choose **User defined**. In the associated text field, type `0.2`.

Heat Flux 1 - Water Jacket

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 In the **Settings** window for **Heat Flux**, type `Heat Flux 1 - Water Jacket` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Water Jacket - External Boundaries**.

4 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

5 In the h text field, type `500`.

6 In the T_{ext} text field, type `25[degC]`.

Heat Flux 2 - Airgap

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 In the **Settings** window for **Heat Flux**, type `Heat Flux 2 - Airgap` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Airgap Heat Flux Boundaries**.

4 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

5 In the h text field, type `50`.

6 In the T_{ext} text field, type `aveop2(T)`.

Periodic Condition 1

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

2 Click the **Select Box** button in the **Graphics** toolbar.

3 Select Boundaries 1, 2, 31, 36, 123, 127, 129, and 132 only.

MULTIPHYSICS

Electromagnetic Heating 1 (emh1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain> Electromagnetic Heating**.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **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 **Curvature factor** text field, type `0.6`.

5 In the **Resolution of narrow regions** text field, type `0.5`.

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

6 Click to expand the **Element Size Parameters** section. 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 `1`.

9 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 **General Studies>Stationary**.

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

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

STUDY 1 - CONVERGENCE WITH NUMBER OF TIME FRAMES

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

2 In the **Settings** window for **Study**, type `Study 1 - Convergence with Number of Time Frames` in the **Label** text field.

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**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Nframes (Number of time frames)	10 30 60 120 180 240	

In order to parameterize the number of time frames solved for it is necessary to disable the Parametric solver.

5 Click to expand the **Advanced Settings** section. From the **Use parametric solver** list, choose **Off**.

6 Select the **Reuse solution from previous step** check box.

Solution 1 (sol1)

While the Heat transfer is heavily influenced by the Magnetic Machinery in this case, the dependence on the coil resistivity and magnetic remanence flux density is only loosely coupled with temperature. For this problem, a Segregated solver is more efficient than the default Fully coupled.

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 Right-click **Study 1 - Convergence with Number of Time Frames>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** and choose **Segregated**.

4 In the **Model Builder** window, expand the **Study 1 - Convergence with Number of Time Frames>Solver Configurations>Solution 1 (sol1)> Stationary Solver 1>Segregated 1** node, then click **Segregated Step**.

5 In the **Settings** window for **Segregated Step**, type `Magnetic Field` in the **Label** text field.

6 Locate the **General** section. In the **Variables** list, choose **External temperature (comp1.ht.TextFace)** and **Temperature (comp1.T)**.

7 Under **Variables**, click **Delete**.

8 Click to expand the **Method and Termination** section. From the **Termination technique** list, choose **Tolerance**.

9 In the **Model Builder** window, under **Study 1 - Convergence with Number of Time Frames>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** right-click **Segregated 1** and choose **Segregated Step**.

10 In the **Settings** window for **Segregated Step**, type `Temperature Field` in the **Label** text field.

11 Locate the **General** section. Under **Variables**, click **Add**.

12 In the **Add** dialog box, in the **Variables** list, choose **External temperature (comp1.ht.TextFace)** and **Temperature (comp1.T)**.

13 Click **OK**.

STUDY 1 - CONVERGENCE WITH NUMBER OF TIME FRAMES

Solver Configurations

1 In the **Model Builder** window, collapse the **Study 1 - Convergence with Number of Time Frames>Solver Configurations** node.

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

The following steps will create an Evaluation group and a plot to inspect the convergence with number of time frames.

RESULTS

Evaluation Group 1

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

Global Evaluation 1

1 Right-click **Evaluation Group 1** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Magnetic Machinery, Rotating, Time Periodic>Mechanical>mmtp.rcon1.Tax_tpavg - Axial torque, time periodic average - N·m**.

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

Expression	Unit	Description
mmtp.rcon1.Tax_tpavg	N*m	Torque

Surface Integration 1

1 In the **Model Builder** window, right-click **Evaluation Group 1** and choose **Integration>Surface Integration**.

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

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

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

Expression	Unit	Description
`mmtp.Qh*L*Nsec`	`W`	`Winding`

5 Right-click **Surface Integration 1** and choose **Duplicate**.

Surface Integration 2

1 In the **Model Builder** window, click **Surface Integration 2**.

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

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

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

Expression	Unit	Description
`mmtp.Qh*L*Nsec`	`W`	`Stator core`

5 Right-click **Surface Integration 2** and choose **Duplicate**.

Surface Integration 3

1 In the **Model Builder** window, click **Surface Integration 3**.

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

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

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

Expression	Unit	Description
`mmtp.Qh*L*Nsec`	`W`	`Magnets`

5 Right-click **Surface Integration 3** and choose **Duplicate**.

Surface Integration 4

1 In the **Model Builder** window, click **Surface Integration 4**.

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

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

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

Expression	Unit	Description
`mmtp.Qh*L*Nsec`	`W`	`Rotor core`

Evaluation Group 1

1 In the **Model Builder** window, click **Evaluation Group 1**.

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

3 From the **Dataset** list, choose **Study 1 - Convergence with Number of Time Frames/ Parametric Solutions 1 (sol2)**.

4 In the **Evaluation Group 1** toolbar, click **=** **Evaluate**.

Convergence with Number of Frames

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

2 In the **Settings** window for **1D Plot Group**, type `Convergence with Number of Frames` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** check box. In the associated text field, type `Electromagnetic loss [W]`.

5 Select the **Two y-axes** check box.

6 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

Table Graph 1

1 Right-click **Convergence with Number of Frames** and choose **Table Graph**.

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

3 From the **Source** list, choose **Evaluation group**.

4 From the **x-axis data** list, choose **Nframes**.

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

6 In the **Columns** list, select **Torque (N*m)**.

7 Locate the **y-Axis** section. Select the **Plot on secondary y-axis** check box.

8 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

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

10 Right-click **Table Graph 1** and choose **Duplicate**.

Table Graph 2

1 In the **Model Builder** window, click **Table Graph 2**.

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

3 In the **Columns** list, choose **Winding (W)**, **Stator core (W)**, **Magnets (W)**, and **Rotor core (W)**.

4 Locate the **y-Axis** section. Clear the **Plot on secondary y-axis** check box.

5 In the **Convergence with Number of Frames** toolbar, click **Plot**.

RESULTS

Convergence with Number of Frames

In the **Model Builder** window, collapse the **Results>Convergence with Number of Frames** node.

STUDY 1 - CONVERGENCE WITH NUMBER OF TIME FRAMES

Update the number of frames and create a new study sweeping over a range of speeds and current levels needed to generate the efficiency map.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, expand the **Study 1 - Convergence with Number of Time Frames** node, then click **Global Definitions>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
Nframes	120	120	Number of time frames

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 **General Studies>Stationary**.

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

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

STUDY 2 - EFFICIENCY MAP

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

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

Parametric Sweep

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

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

3 From the **Sweep type** list, choose **All combinations**.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
w_rot (Shaft speed)	`range(1200,3600,12000)`	`rpm`

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Ipk (Phase current peak value)	`range(3,6.75,30)`	A

8 In the table, click to select the cell at row number 2 and column number 3.

9 Locate the **Advanced Settings** section. Select the **Reuse solution from previous step** check box.

Solution 9 (sol9)

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

2 In the **Model Builder** window, expand the **Solution 9 (sol9)** node.

3 In the **Model Builder** window, expand the **Study 2 - Efficiency Map>Solver Configurations>Solution 9 (sol9)>Stationary Solver 1** node.

4 Right-click **Study 2 - Efficiency Map>Solver Configurations>Solution 9 (sol9)>Stationary Solver 1** and choose **Segregated**.

5 In the **Model Builder** window, expand the **Study 2 - Efficiency Map>Solver Configurations>Solution 9 (sol9)>Stationary Solver 1>Segregated 1** node, then click **Segregated Step**.

6 In the **Settings** window for **Segregated Step**, type `Magnetic Field` in the **Label** text field.

7 Locate the **General** section. In the **Variables** list, choose **External temperature (comp1.ht.TextFace)** and **Temperature (comp1.T)**.

8 Under **Variables**, click **Delete**.

9 Locate the **Method and Termination** section. From the **Termination technique** list, choose **Tolerance**.

10 In the **Model Builder** window, under **Study 2 - Efficiency Map>Solver Configurations> Solution 9 (sol9)>Stationary Solver 1** right-click **Segregated 1** and choose **Segregated Step**.

11 In the **Settings** window for **Segregated Step**, type `Temperature Field` in the **Label** text field.

12 Locate the **General** section. Under **Variables**, click **Add**.

13 In the **Add** dialog box, in the **Variables** list, choose **External temperature (comp1.ht.TextFace)** and **Temperature (comp1.T)**.

14 Click **OK**.

STUDY 2 - EFFICIENCY MAP

Solver Configurations

1 In the **Model Builder** window, collapse the **Study 2 - Efficiency Map>Solver Configurations** node.

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

The following steps duplicate and modify the already existing evaluation group and use it to generate the efficiency map.

RESULTS

Evaluation Group 1

In the **Model Builder** window, under **Results** right-click **Evaluation Group 1** and choose **Duplicate**.

Evaluation Group 2

1 In the **Model Builder** window, click **Evaluation Group 2**.

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

3 From the **Dataset** list, choose **Study 2 - Efficiency Map/Solution 9 (sol9)**.

Global Evaluation 1

1 In the **Model Builder** window, expand the **Evaluation Group 2** node.

2 Right-click **Global Evaluation 1** and choose **Duplicate**.

Global Evaluation 2

1 In the **Model Builder** window, click **Global Evaluation 2**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
mmtp.rcon1.Tax_tpavg*w_rot*2*pi	W	Shaft power

Evaluation Group 2

1 In the **Model Builder** window, click **Evaluation Group 2**.

2 In the **Settings** window for **Evaluation Group**, locate the **Transformation** section.

3 From the **Transformation type** list, choose **General**.

4 Select the **Keep child nodes** check box.

5 In the **Expression** text field, type `gev2/(gev2+int1+int2+int3+int4)`.

6 In the **Column header** text field, type `Efficiency`.

7 In the **Evaluation Group 2** toolbar, click **Evaluate**.

EVALUATION GROUP 2

1 Go to the **Evaluation Group 2** window.

2 Click **Table Contour** in the window toolbar.

RESULTS

Table Contour 1

1 In the **Model Builder** window, under **Results>2D Plot Group 6** click **Table Contour 1**.

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

3 From the **y-axis column** list, choose **Torque (N*m)**.

4 Right-click **Results>2D Plot Group 6>Table Contour 1** and choose **Duplicate**.

Table Contour 2

1 In the **Model Builder** window, click **Table Contour 2**.

2 In the **Settings** window for **Table Contour**, locate the **Coloring and Style** section.

3 From the **Contour type** list, choose **Line**.

4 Select the **Level labels** check box.

5 In the **Precision** text field, type `2`.

6 From the **Label color** list, choose **Black**.

7 Click **Change Color Table**.

8 In the **Color Table** dialog box, select **Rainbow>RainbowDark** in the tree.

9 Click **OK**.

10 In the **Settings** window for **Table Contour**, locate the **Coloring and Style** section.

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

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

Motor Efficiency Map

1 In the **Model Builder** window, under **Results** click **2D Plot Group 6**.

2 In the **Settings** window for **2D Plot Group**, type `Motor Efficiency Map` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **x-axis label** check box. In the associated text field, type `Speed (rpm)`.

5 In the **Motor Efficiency Map** toolbar, click **Plot**.

Evaluation Group 2

In the **Model Builder** window, collapse the **Results>Evaluation Group 2** node.

Motor Efficiency Map

1 In the **Model Builder** window, collapse the **Results>Motor Efficiency Map** node.

The remaining steps are rather repetitive and only needed if you want to reproduce Figure 4 and Figure 5 in the Results and Discussion section.

2 In the **Model Builder** window, expand the **Results** node.

Surface 1

1 In the **Model Builder** window, expand the **Results>Temperature (ht) 1** node, then click **Surface 1**.

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

3 From the **Dataset** list, choose **Study 2 - Efficiency Map/Solution 9 (sol9)**.

4 From the **Solution parameters** list, choose **Manual**.

5 From the **Parameter value (w_rot (rpm))** list, choose **1200**.

6 From the **Parameter value (Ipk (A))** list, choose **3**.

7 Locate the **Expression** section. From the **Unit** list, choose **degC**.

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

Annotation 1

1 In the **Model Builder** window, right-click **Temperature (ht) 1** and choose **Annotation**.

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

3 From the **Dataset** list, choose **Study 2 - Efficiency Map/Solution 9 (sol9)**.

4 From the **Parameter value (w_rot (rpm))** list, choose **1200**.

5 From the **Parameter value (Ipk (A))** list, choose **3**.

6 Locate the **Annotation** section. In the **Text** text field, type `speed = eval(w_rot,rpm,5)  Ipk = eval(Ipk,A,3)`.

7 Locate the **Position** section. In the **X** text field, type `-16`.

8 In the **Y** text field, type `6`.

9 Locate the **Coloring and Style** section. Clear the **Show point** check box.

10 From the **Background color** list, choose **From theme**.

11 Select the **Show frame** check box.

Annotation 1, Surface 1

1 In the **Model Builder** window, under **Results>Temperature (ht) 1**, Ctrl-click to select **Surface 1** and **Annotation 1**.

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

Surface 2

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

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

3 From the **Parameter value (Ipk (A))** list, choose **30**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Translation 1

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

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

3 In the **y** text field, type `30`.

Translation 1

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

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

3 In the **y** text field, type `30`.

Annotation 2

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

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

3 From the **Parameter value (Ipk (A))** list, choose **30**.

Annotation 2, Surface 2

1 In the **Model Builder** window, under **Results>Temperature (ht) 1**, Ctrl-click to select **Surface 2** and **Annotation 2**.

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

Surface 3

1 In the **Model Builder** window, click **Surface 3**.

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

3 From the **Parameter value (w_rot (rpm))** list, choose **12000**.

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 60.

Annotation 3

1 In the **Model Builder** window, under **Results>Temperature (ht) 1** click **Annotation 3**.

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

3 From the **Parameter value (w_rot (rpm))** list, choose **12000**.

Translation 1

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

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

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

Annotation 3, Surface 3

1 In the **Model Builder** window, under **Results>Temperature (ht) 1**, Ctrl-click to select **Surface 3** and **Annotation 3**.

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

Surface 4

1 In the **Model Builder** window, click **Surface 4**.

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

3 From the **Parameter value (Ipk (A))** list, choose **3**.

Translation 1

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

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

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

Annotation 4

1 In the **Model Builder** window, under **Results>Temperature (ht) 1** click **Annotation 4**.

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

3 From the **Parameter value (Ipk (A))** list, choose **3**.

Translation 1

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

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

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

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

RESULTS

Temperature (ht) 1

1 In the **Model Builder** window, collapse the **Results>Temperature (ht) 1** node.

2 In the **Model Builder** window, click **Temperature (ht) 1**.

3 In the **Settings** window for **2D Plot Group**, click to expand the **Title** section.

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

5 In the **Title** text area, type `Temperature fields of four operating points`.

6 Clear the **Parameter indicator** text field.

7 Locate the **Color Legend** section. Select the **Show units** check box.

8 In the **Temperature (ht) 1** toolbar, click **Plot**.

9 Right-click **Temperature (ht) 1** and choose **Duplicate**.

Electromagnetic Loss

1 In the **Model Builder** window, under **Results** click **Temperature (ht) 1.1**.

2 In the **Settings** window for **2D Plot Group**, type `Electromagnetic Loss` in the **Label** text field.

3 Click to expand the **Title** section. In the **Title** text area, type `Electromagnetic loss distribution of four operating points`.

Surface 1

1 In the **Model Builder** window, expand the **Electromagnetic Loss** 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>Heating and losses>mmtp.Qh - Volumetric loss density, electromagnetic - W/m^3.

3 Locate the **Coloring and Style** section. Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Rainbow>Rainbow** in the tree.

5 Click **OK**.

Surface 2

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

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **mmtp.Qh - Volumetric loss density, electromagnetic - W/m^3**.

Surface 3

1 In the **Model Builder** window, click **Surface 3**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **mmtp.Qh - Volumetric loss density, electromagnetic - W/m^3**.

Surface 4

1 In the **Model Builder** window, click **Surface 4**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **mmtp.Qh - Volumetric loss density, electromagnetic - W/m^3**.

Electromagnetic Loss

1 In the **Model Builder** window, collapse the **Results>Electromagnetic Loss** node.

2 In the **Model Builder** window, click **Electromagnetic Loss**.

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

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