Model created in COMSOL Multiphysics 6.4

Induction Hardening of a Cylindrical Pin

Introduction

Induction hardening is a heat treatment process that is used to increase hardness and wear resistance of steel parts. The process uses a copper coil with a high frequency alternating current to induce eddy currents on the surface layer of the part. These eddy currents heat the material rapidly above the austenitization temperature, and the part is then rapidly quenched in water, oil, or similar. This typically produces a hard and durable martensitic surface, while retaining the softer base composition inside.

This example shows how to perform an induction hardening simulation for a small steel pin. The pin is made from a hypoeutectoid ferritic–pearlitic steel and is positioned inside a multiturn coil, where it is rapidly heated and then quickly cooled using a water spray.

Model Definition

This section describes the various aspects involved in setting up and performing an induction hardening simulation.

GEOMETRY

The pin is 30 mm in length and 10 mm in diameter, and has a 1 mm chamfer at its ends; see Figure 1.

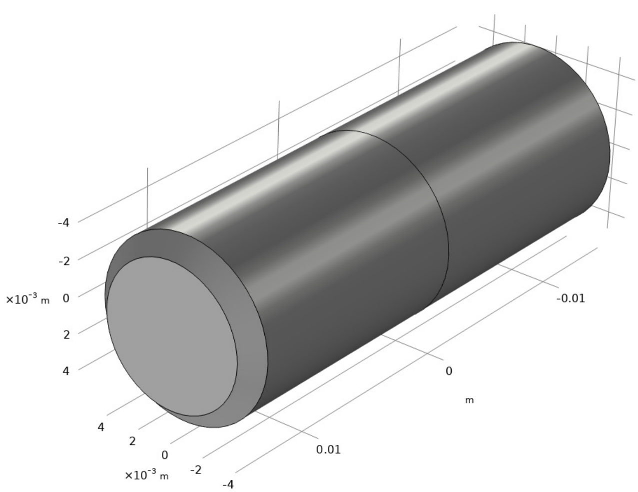

Figure 1: The pin used in the simulation.

As the pin is axisymmetric as well as symmetric across its center, half the pin is modeled in 2D axisymmetry.

MATERIAL PROPERTIES

Material properties for the phases austenite, ferrite, pearlite, bainite, and martensite are imported from JMatPro® (Ref. 1). They include thermal properties, mechanical properties, and electrical properties, and they are in general temperature dependent. Elastoplastic properties additionally depend on plastic strain and plastic strain rate. Magnetic permeability is treated separately in JMatPro®. It is imported into COMSOL Multiphysics as a single, compound material with averaged properties, rather than as separate phase-specific materials.

PHASE TRANSFORMATIONS AND INITIAL COMPOSITION

Phase-transformation data for austenite decomposition are imported from JMatPro® (Ref. 1). Four phase transformation models are automatically configured to represent the decomposition into ferrite, pearlite, bainite, and martensite, respectively. For austenitization, a simple linear phase-transformation model is used. This model is based on the simple idea that the rate of formation of the destination phase (austenite) is proportional to the heating rate. The upper and lower temperature limits T_u and T_l in the denominator are taken as 900 and 723 degrees Celsius, respectively. The rate equation is given by

$$\dot{\xi}^{d} = \frac{\dot{T}}{T_u - T_l}$$

For a more accurate description of the austenitization process, a more sophisticated phase-transformation model is required, but for the present example the linear model will suffice. The base (starting) composition of the steel is taken to be 50% ferrite and 50% pearlite.

THERMAL BOUNDARY CONDITIONS

The surface of the pin will transfer heat to its surroundings during the induction hardening process. The model considers three mechanisms for heat transfer: Convection to the surrounding air, thermal radiation to the surrounding air, and convection to the water during the water spray cooling.

Convection to Air

For simplicity, this mode of heat transfer is modeled using a constant heat transfer coefficient of 15 W/(m^2·K), considering the ambient air to be 20 degrees Celsius.

Thermal Radiation

When the pin is heated it becomes increasingly important to account for thermal radiation. Here, the emissivity of the steel surface is 0.8.

Water Spray Cooling

The water spray cooling is modeled using a temperature-dependent heat transfer coefficient; see Figure 2. The water temperature is taken to be 60 degrees Celsius, and this convection mechanism is active only when the induction current has been turned off. This represents rapidly removing the inductor from the pin, and spraying water on its surface.

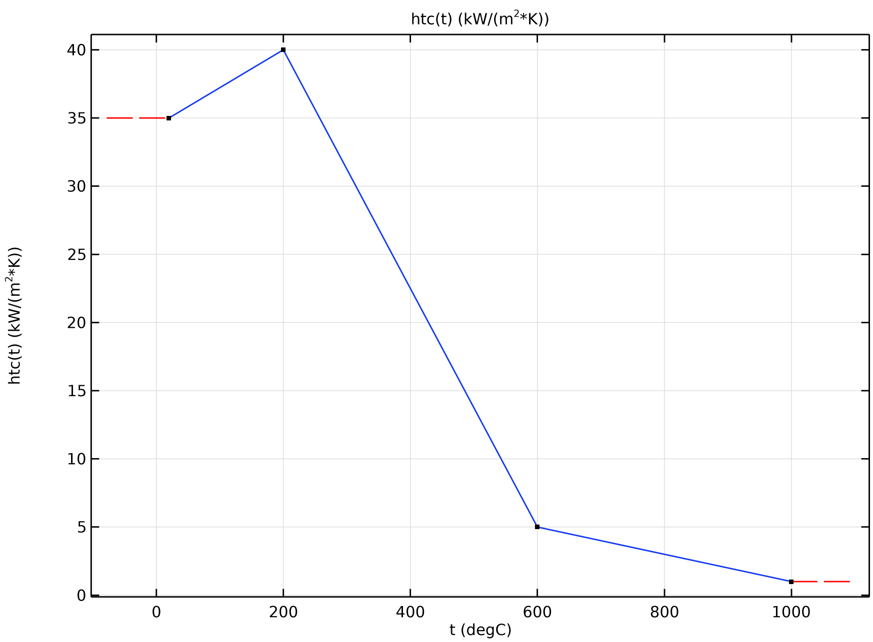

Figure 2: The temperature dependent heat transfer coefficient for the water spray cooling.

MECHANICAL BOUNDARY CONDITIONS

Because only half the pin is modeled, a symmetry boundary condition is applied at the center plane.

ELECTROMAGNETIC BOUNDARY CONDITIONS AND LOADING

Because only half the pin is modeled, an antisymmetry condition is applied to the magnetic flux density across the geometric symmetry plane. The computational domain is chosen large compared to the pin itself, and it is magnetically insulated along its perimeter.

The twenty-turn induction coil is made from copper and the alternating current is increasing linearly over half a second to a peak value of 175 A and then down linearly again for a total excitation time of one second. The frequency of the induction current is 120 kHz.

Results and Discussion

The purpose of using induction heating compared to, for example, a furnace, is that it is possible to essentially heat only the surface of the part. This means that the interior of the part remains at its ferritic–pearlitic base composition, and only the surface is austenitized. After quenching the part, here using water spray cooling, the austenitic surface transforms into hard martensite; see Figure 3. Because of the lower density of martensite compared to austenite, the phase transformation produces volumetric expansion, and this results in compressive stresses near the surface. Figure 4 shows the axial stress component after hardening. Stresses are compressive near the surface, and balanced by tensile stresses on the inside of the pin. From a durability standpoint, compressive residual surface stresses are usually desired.

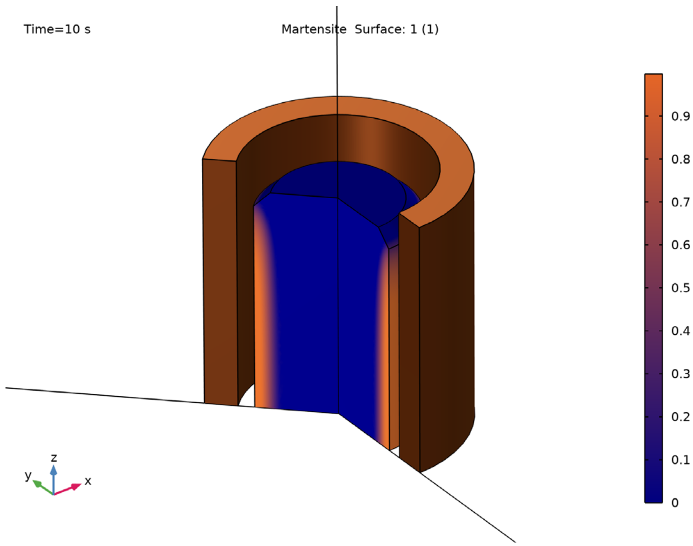

Figure 3: Phase fraction of martensite after hardening.

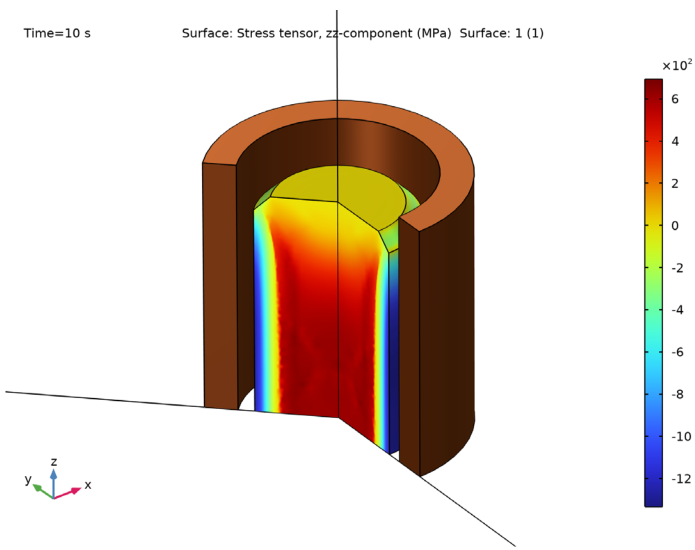

Figure 4: Stress in the Z direction after hardening.

Notes About the COMSOL Implementation

- When material properties are imported from JMatPro®, the relative permeability is not provided per metallurgical phase. Instead, it is imported as an effective quantity for the material. It is therefore merged into the compound material after import.
- Because the induction frequency is rather high (125 kHz), it may be necessary to mesh the surface of the pin even finer to adequately resolve the skin effect.
- Phase-transformation data is imported by selecting **Import Phase Transformations** in the physics interface's context menu.
- Phase material properties are imported by selecting **Import Materials** from the **Materials** context menu under **Global Definitions** or under **Materials** at the component level (not available in 0D).

Reference

1. Sente Software, Ltd., United Kingdom.

Application Library path: `Metal_Processing_Module/Induction_Hardening/induction_hardening_of_a_cylindrical_pin`

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 Axisymmetric**.

2 In the **Select Physics** tree, select **Heat Transfer** > **Metal Processing** > **Induction Hardening**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics** > **Frequency–Transient**.

6 Click **Done**.

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 `0.005`.

4 In the **Height** text field, type `0.015`.

5 In the **Width** text field, type `0.1`.

6 In the **Height** text field, type `0.1`.

Rectangle 2 (r2)

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 `0.005`.

4 In the **Height** text field, type `0.015`.

Chamfer 1 (cha1)

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

2 On the object **r2**, select Point 3 only.

3 In the **Settings** window for **Chamfer**, locate the **Distance** section.

4 In the **Distance from vertex** text field, type `0.001`.

Rectangle 3 (r3)

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 `0.002`.

4 In the **Height** text field, type `0.017`.

5 Locate the **Position** section. In the **r** text field, type `0.006`.

Form Union (fin)

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

Pin

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

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

3 On the object **fin**, select Domain 1 only.

Coil

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

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

3 On the object **fin**, select Domain 3 only.

Air

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

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

3 On the object **fin**, select Domain 2 only.

Pin Surface

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

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

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

4 On the object **fin**, select Boundaries 4, 6, and 7 only.

5 In the **Label** text field, type `Pin Surface`.

Symmetry Plane

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

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

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

4 On the object **fin**, select Boundaries 2, 8, 10, and 13 only.

5 In the **Label** text field, type `Symmetry Plane`.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in** > **Air**.

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

MATERIALS

Air (mat1)

1 Select Domain 3 only.

2 Click the **Zoom In** button in the **Graphics** toolbar.

3 Select Domain 2 only.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **Built-in** > **Copper**.
3 Click the **Add to Component** button in the window toolbar.
4 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Copper (mat2)

1 Select Domain 3 only.
2 In the **Materials** toolbar, click **Import Materials** and choose **Import Materials**.
3 In the **Import Materials** dialog, click **Browse**.
4 Browse to the model's Application Libraries folder and double-click the file `induction_hardening_of_a_cylindrical_pin_JMatPro_general_steel.xml`.
5 Click **OK**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
tEnd	1[s]	1 s	End of induction

Interpolation 1 (int1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global** > **Interpolation**.
2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
3 In the **Function name** text field, type `htc`.
4 In the table, enter the following settings:

t	f(t)
20	35
200	40
600	5
1000	1

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

Function	Unit
htc	kW/(m^2*K)

6 In the **Argument** table, enter the following settings:

Argument	Unit
t	degC

Interpolation 2 (int2)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global** > **Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `current`.

4 In the table, enter the following settings:

t	f(t)
0	0
tEnd/2	175
tEnd	1

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

Function	Unit
current	A

6 In the **Argument** table, enter the following settings:

Argument	Unit
t	s

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 **Pin**.

Heat Flux 1

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

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

3 From the **Selection** list, choose **Pin Surface**.

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

5 In the h text field, type `htc(T)*(t>tEnd)`.

6 In the T_{ext} text field, type `60[degC]`.

Heat Flux 2

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

2 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

3 From the **Flux type** list, choose **Convective heat flux**.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Pin Surface**.

5 Locate the **Heat Flux** section. In the h text field, type `15`.

6 In the T_{ext} text field, type `20[degC]`.

Surface-to-Ambient Radiation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Surface-to-Ambient Radiation**.

2 In the **Settings** window for **Surface-to-Ambient Radiation**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Pin Surface**.

4 Locate the **Surface-to-Ambient Radiation** section. From the ε list, choose **User defined**. In the associated text field, type `0.8`.

5 In the T_{amb} text field, type `20[degC]`.

SOLID MECHANICS (SOLID)

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

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

3 From the **Selection** list, choose **Pin**.

Linear Elastic Material 1

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

Plasticity 1

1 In the **Physics** toolbar, click **Attributes** and choose **Plasticity**.

2 In the **Settings** window for **Plasticity**, locate the **Plasticity Model** section.

3 Find the **Isotropic hardening model** subsection. From the list, choose **Hardening function**.

Symmetry Plane 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry Plane**.

2 Select Boundary 2 only.

AUSTENITE DECOMPOSITION (AUDC)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Austenite Decomposition (audc)**.

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

3 From the **Selection** list, choose **Pin**.

4 Locate the **Solid Mechanics** section. Select the **Enable phase plasticity** checkbox.

5 From the **Thermal strain formulation** list, choose **Density based**.

6 Locate the **Material Properties** section. Click **Create Compound Material** in the upper-right corner of the section.

MATERIALS

Air (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Air (mat1)**.

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

3 From the **Selection** list, choose **Air**.

Copper (mat2)

1 In the **Model Builder** window, click **Copper (mat2)**.

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

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

Compound Material (mat9)

1 In the **Model Builder** window, click **Compound Material (mat9)**.

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

3 From the **Selection** list, choose **Pin**.

General Steel, Effective Properties (mat8)

In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** right-click **General Steel, Effective Properties (mat8)** and choose **Merge Into** > **Compound Material (mat9)**.

AUSTENITE DECOMPOSITION (AUDC)

Austenite

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Austenite Decomposition (audc)** click **Austenite**.

2 In the **Settings** window for **Metallurgical Phase**, locate the **Initial Phase Fraction** section.

3 In the ξ_0 text field, type `0`.

4 Locate the **Phase Material** section. From the **Phase material** list, choose **General Steel, Austenite (mat3)**.

5 Locate the **Electromagnetic Properties** section. From the ε_r list, choose **User defined**. Locate the **Mechanical Properties** section. From the **Isotropic hardening model** list, choose **Hardening function**.

Ferrite

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

2 In the **Settings** window for **Metallurgical Phase**, locate the **Initial Phase Fraction** section.

3 In the ξ_0 text field, type `0.5`.

4 Locate the **Phase Material** section. From the **Phase material** list, choose **General Steel, Ferrite (mat4)**.

5 Locate the **Electromagnetic Properties** section. From the ε_r list, choose **User defined**. Locate the **Mechanical Properties** section. From the **Isotropic hardening model** list, choose **Hardening function**.

Pearlite

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

2 In the **Settings** window for **Metallurgical Phase**, locate the **Initial Phase Fraction** section.

3 In the ξ_0 text field, type `0.5`.

4 Locate the **Phase Material** section. From the **Phase material** list, choose **General Steel, Pearlite (mat5)**.

5 Locate the **Electromagnetic Properties** section. From the ε_r list, choose **User defined**. Locate the **Mechanical Properties** section. From the **Isotropic hardening model** list, choose **Hardening function**.

Bainite

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

2 In the **Settings** window for **Metallurgical Phase**, locate the **Phase Material** section.

3 From the **Phase material** list, choose **General Steel, Bainite (mat6)**.

4 Locate the **Electromagnetic Properties** section. From the ε_r list, choose **User defined**. Locate the **Mechanical Properties** section. From the **Isotropic hardening model** list, choose **Hardening function**.

Martensite

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

2 In the **Settings** window for **Metallurgical Phase**, locate the **Phase Material** section.

3 From the **Phase material** list, choose **General Steel, Martensite (mat7)**.

4 Locate the **Electromagnetic Properties** section. From the ε_r list, choose **User defined**. Locate the **Mechanical Properties** section. From the **Isotropic hardening model** list, choose **Hardening function**.

General Steel, Austenite to Ferrite

1 In the **Model Builder** window, right-click **Austenite Decomposition (audc)** and choose **Import Phase Transformations**.

2 Browse to the model's Application Libraries folder and double-click the file `induction_hardening_of_a_cylindrical_pin_JMatPro_general_steel.xml`.

3 In the **Settings** window for **Phase Transformation**, locate the **Phase Transformation** section.

4 From the ξ^s list, choose **Austenite**.

5 From the ξ^d list, choose **Ferrite**.

General Steel, Austenite to Pearlite

1 In the **Model Builder** window, click **General Steel, Austenite to Pearlite**.

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

3 From the ξ^s list, choose **Austenite**.

4 From the ξ^d list, choose **Pearlite**.

General Steel, Austenite to Bainite

1 In the **Model Builder** window, click **General Steel, Austenite to Bainite**.

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

3 From the ξ^s list, choose **Austenite**.

4 From the ξ^d list, choose **Bainite**.

General Steel, Austenite to Martensite

1 In the **Model Builder** window, click **General Steel, Austenite to Martensite**.

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

3 From the ξ^s list, choose **Austenite**.

4 From the ξ^d list, choose **Martensite**.

Austenitization

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

2 In the **Settings** window for **Phase Transformation**, type `Austenitization` in the **Label** text field.

3 Locate the **Phase Transformation** section. From the ξ^s list, choose **Ferrite**.

4 From the ξ^d list, choose **Austenite**.

5 From the **Phase transformation model** list, choose **Linear**.

6 In the T_1 text field, type `723[degC]`.

Additional Source Phase 1

1 In the **Physics** toolbar, click **Attributes** and choose **Additional Source Phase**.

2 In the **Settings** window for **Additional Source Phase**, locate the **Additional Source Phase** section.

3 From the ξ^s list, choose **Pearlite**.

MAGNETIC FIELDS (MF)

Initial Values 1

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

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 Specify the **A** vector as

`1e-8*sqrt(r^2+z^2)`	phi

Coil 1

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

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

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

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

5 In the I_{coil} text field, type `current(t)`.

6 Locate the **Homogenized Conductor** section. In the N text field, type `20`.

Ampère's Law in Solids 1

1 In the **Physics** toolbar, click **Domains** and choose **Ampère's Law in Solids**.
2 In the **Settings** window for **Ampère's Law in Solids**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **Pin**.

Symmetry Plane 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry Plane**.
2 In the **Settings** window for **Symmetry Plane**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Symmetry Plane**.
4 Locate the **Symmetry Plane** section. From the **Symmetry type for the magnetic flux density** list, choose **Antisymmetry**.

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.
2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **Coil**.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 In the **Number of elements** text field, type 6.
4 Select Boundary 11 only.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 In the **Number of elements** text field, type 30.
4 Select Boundary 12 only.

Distribution 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Distribution**.
2 Select Boundary 7 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type 60.

Distribution 2

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

2 Select Boundary 4 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `10`.

Size 1

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

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

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

4 From the **Selection** list, choose **Pin**.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.

Boundary Layers 1

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

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

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

4 From the **Selection** list, choose **Pin**.

Boundary Layer Properties

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

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

3 From the **Selection** list, choose **Pin Surface**.

4 Locate the **Layers** section. From the **Thickness specification** list, choose **All layers**.

5 In the **Total thickness** text field, type `0.001`.

Free Triangular 1

1 In the **Mesh** toolbar, click **Free Triangular**.

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

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

4 From the **Selection** list, choose **Air**.

STUDY 1

Step 1: Frequency–Transient

1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency–Transient**.

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

3 In the **Output times** text field, type `range(0,0.1,tEnd*10)`.

4 In the **Frequency** text field, type `120[kHz]`.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Compile Equations: Frequency–Transient**.

3 In the **Settings** window for **Compile Equations**, locate the **Study and Step** section.

4 Select the **Split complex variables in real and imaginary parts** checkbox.

5 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** right-click **Time-Dependent Solver 1** and choose **Segregated**.

6 In the **Settings** window for **Segregated**, locate the **General** section.

7 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.

8 In the **Maximum number of iterations** text field, type `50`.

9 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** > **Segregated 1** node, then click **Segregated Step**.

10 In the **Settings** window for **Segregated Step**, locate the **General** section.

11 In the **Variables** list, choose **Phase Fractions (comp1.audc.phasefractions)**, **Phase Transformation Strains (comp1.ptstr1.ptstrains)**, **Time at Previous Step (comp1.ptstr1.t_old)**, **Temperature (comp1.T)**, and **Displacement Field (comp1.u)**.

12 Under **Variables**, click **Delete**.

13 Click to expand the **Method and Termination** section. From the **Nonlinear method** list, choose **Backtracking (Newton)**.

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

15 From the **Termination technique** list, choose **Iterations or tolerance**.

16 In the **Number of iterations** text field, type `100`.

17 In the **Tolerance factor** text field, type `1`.

18 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** right-click **Segregated 1** and choose **Segregated Step**.

19 In the **Settings** window for **Segregated Step**, locate the **General** section.

20 Under **Variables**, click **Add**.

21 In the **Add** dialog, in the **Variables** list, choose
Phase Fractions (comp1.audc.phasefractions),
Phase Transformation Strains (comp1.ptstr1.ptstrains),
Time at Previous Step (comp1.ptstr1.t_old), **Temperature (comp1.T)**, and
Displacement Field (comp1.u).

22 Click **OK**.

23 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.

24 From the **Nonlinear method** list, choose **Automatic (Newton)**.

25 From the **Termination technique** list, choose **Iterations or tolerance**.

26 In the **Number of iterations** text field, type 10.

27 In the **Tolerance factor** text field, type 1.

28 In the **Study** toolbar, click **= Compute**.