Model created in COMSOL Multiphysics 6.4

Quenching of a Steel Billet

Introduction

Quenching is a heat treating process that is used to tailor the microstructure, and to control distortions and residual stresses in steel components. If distortions can be minimized, postquenching manufacturing operations such as grinding can be avoided. From an endurance standpoint, compressive residual stresses at the surface of a component can be beneficial because the propensity for fatigue failure is reduced. A tendency is to try to use steel components as heat treated. This preserves beneficial compressive stresses on the surface, and reduces overall manufacturing costs.

In this example, a steel billet is considered. The billet is first heated to 900°C, and then quenched in oil. As the temperature decreases, the austenite decomposes into a combination of ferrite, pearlite, bainite, and martensite. The model shows how to define the temperature-dependent metallurgical phase transformations that are involved in this process, and how to compute the heterogeneous phase composition in the billet. During quenching, phase transformation strains produce stresses and deformations. The model shows how to compute these stresses and deformations by coupling the temperature-dependent phase transformations to an elastoplastic analysis. Effects such as traditional plasticity as well as transformation induced plasticity (TRIP) are included.

Model Definition

The steel billet is a solid cylinder that is 20 cm in length and with a radius of 2 cm. The billet is quenched in oil, uniformly across its boundary, through a temperature-dependent heat transfer coefficient. Because of symmetries, only half the billet is considered, in 2D axisymmetry. The billet is shown in Figure 1. Thermal and mechanical boundary conditions are discussed below.

MATERIAL PROPERTIES

The material properties of the steel billet are temperature-dependent, and also phase dependent. The Austenite Decomposition physics interface automatically averages these properties into effective properties that define a compound material. The compound material is used in the thermal and mechanical analyses.

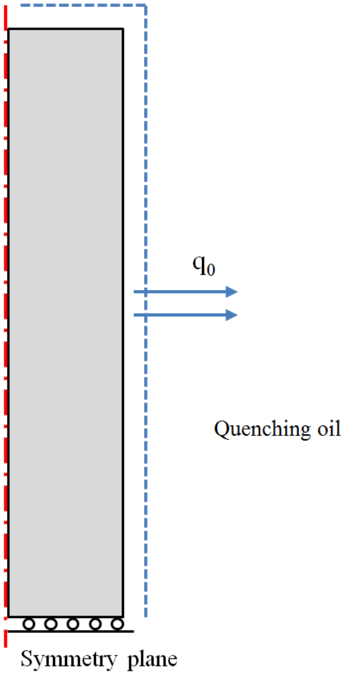

Figure 1: The axisymmetric model of the steel billet.

PHASE TRANSFORMATION ANALYSIS

During cooling, the austenite can decompose into a combination of ferrite, pearlite, bainite, and martensite. The phase transformation into martensite is displacive and described by the Koistinen–Marburger model. The model states that the amount of martensite formed at the expense of austenite depends on the fraction of available austenite, and under-cooling below the so-called martensite start temperature M_s. On differential form, the model is given by

$$\dot{\xi}^{d} = -\xi^{s}\beta\dot{T}$$

where the rate at which the destination phase (martensite) forms is proportional to the temperature rate and the instantaneous fraction of the source phase (austenite), through the Koistinen–Marburger coefficient β. Note that martensite only forms during cooling, meaning that the temperature rate must be negative. The remaining diffusive phase transformations are modeled using the Leblond–Devaux model. This model is characterized by a contributing term that is proportional to the available fraction of the source phase, and a retardation term that is proportional to the current fraction of formed destination phase. The proportionality is given by two temperature-dependent functions K and L.

$$\dot{\xi}^{\mathrm{d}} = K(T)\xi^{\mathrm{s}} - L(T)\xi^{\mathrm{d}}$$

The parameters that are required to describe the austenite decomposition into ferrite, pearlite, and bainite are given in Table 1, Table 2, and Table 3 below.

TABLE 1: AUSTENITE TO FERRITE, TEMPERATURE-DEPENDENT FUNCTIONS.

Temperature (°C)	**K (1/s)**	**L (1/s)**
450	0	0
620	0.005	0.001
750	0	0

TABLE 2: AUSTENITE TO PEARLITE, TEMPERATURE-DEPENDENT FUNCTIONS.

Temperature (°C)	**K (1/s)**	**L (1/s)**
450	0	0
550	0.015	0.001
750	0	0

TABLE 3: AUSTENITE TO BAINITE, TEMPERATURE-DEPENDENT FUNCTIONS.

Temperature (°C)	**K (1/s)**	**L (1/s)**
450	0	0
620	0.005	0.001
750	0	0

Martensite forms at the expense of the available fraction of source phase (austenite), and the two parameters that define this phase transformation are given in Table 4.

TABLE 4: AUSTENITE TO MARTENSITE PARAMETERS.

Parameter	**Value**
M_{s}	300°C
β	0.011 /K

THERMAL ANALYSIS

The heat transport in the bar is described by the heat equation:

$$\rho C_p \dot{T} + \nabla \cdot (-k \nabla T) = Q$$

where T is the temperature, k represents the thermal conductivity, ρ denotes the density, C_p denotes the specific heat capacity, and Q is a heat source. The thermal conductivity, the density, and the specific heat capacity are in general temperature dependent, but in the presence of metallurgical phase transformations, they also depend on the current phase composition. For example, the thermal conductivity of austenite is different from that of ferrite, and as the phase fractions evolve, so will the thermal conductivity of the compound material. In the present thermal analysis, phase transformation latent heat is neglected so that $Q = 0$. The densities, specific heat capacities and heat conductivities of the individual metallurgical phases are given in Table 5.

TABLE 5: TEMPERATURE-DEPENDENT THERMAL MATERIAL PROPERTIES.

Temperature (°C)	**ρ (kg/m^3)**	**C_p (J/(kg·K))**	**k (W/(m·K))**
Austenite			
0	7930	520	15
300		560	20
600		590	22
900		620	25
Ferrite, Pearlite, Bainite			
0	7850	480	50
300		570	42
600		640	35
900		700	26
Martensite			
0	7850	480	44
300		570	38
600		640	30
900		650	24

In the model, it is assumed that ferrite, pearlite, and bainite share thermal properties. Furthermore, from a thermal diffusivity standpoint, it is assumed that the densities of the individual phases are temperature independent.

Boundary Conditions

The quenching oil is not modeled explicitly, but it is replaced by a temperature-dependent heat transfer coefficient h that is used to prescribe a heat flux as

$$q_0 = h(T)(T_{\text{ext}} - T)$$

where T_{ext} = 80°C is the temperature of the quenching oil. The heat transfer properties of the quenching oil are shown in Table 6.

TABLE 6: HEAT-TRANSFER COEFFICIENT OF THE QUENCHING OIL.

Temperature (°C)	h (W/(m^2·K))
0	200
300	200
500	2800
650	750
1300	750

MECHANICAL ANALYSIS

The quenching process is time dependent, but from a structural-mechanics point of view it is quasi static, and modeled as such. Stresses and strains are computed using material properties of the compound material defined by the phase composition and the constitutive behavior of the individual phases.

Material Properties

As in thermal analysis, the mechanical analysis involves material properties that are temperature as well as phase composition dependent. In this model, the elastoplastic behavior of the individual metallurgical phases is taken to be linear elastic with linear hardening. The mechanical material properties are shown in Table 7. The linear elastic behavior is given by the Young's moduli (E) and Poisson's ratios (ν) of the phases, and the plastic behavior is given by initial yield stresses (σ_{ys0}) and isotropic hardening moduli (h). In this model, the elastic behavior is assumed to be equal between phases. Note that the secant coefficients of thermal expansion (α) are not averaged into a compound material property, but are instead used to compute the thermal strain tensor of each metallurgical phase. The thermal strain tensors are averaged into a thermal strain of the compound material.

TABLE 7: TEMPERATURE-DEPENDENT MECHANICAL MATERIAL PROPERTIES.

Temperature (°C)	E (GPa)	ν	σ_{ys0} (MPa)	h (GPa)	α (1/K)
Austenite					
0	210	0.3	200	1	$22 \cdot 10^{-6}$
300	180		135	15	
600	165		40	11	

TABLE 7: TEMPERATURE-DEPENDENT MECHANICAL MATERIAL PROPERTIES.

Temperature (°C)	**E** (GPa)	ν	σ_{ys0} (MPa)	**h** (GPa)	α (1/K)
900	120		36	0.6	
Ferrite, Pearlite, Bainite					$15 \cdot 10^{-6}$
0	210	0.3	400	1	
300	180		200	15	
600	165		150	11	
900	120		35	0.6	
Martensite					$14 \cdot 10^{-6}$
0	210	0.3	1600	1	
300	180		1500	15	
600	165		1400	11	
900	120		100	0.6	

To complete the description of the phase properties, a volume reference temperature T_{ref} has to be defined for each metallurgical phase. The choice of volume reference temperature is to an extent arbitrary. In this model, the heating stage (austenitization) is not considered explicitly, so the volume reference temperature is set to the austenitization temperature (900°C). This means that the billet is strain free at this temperature. To account for the strains that follow from thermal expansion and austenitization of the (unknown) base phase composition, an initial strain is applied.

Boundary Conditions

Only half the billet is modeled, and a displacement symmetry boundary condition is applied to the midplane.

Transformation Induced Plasticity (TRIP)

In general, phase transformations occur while the material is subjected to a mechanical stress. This gives rise to so-called transformation induced plasticity, or TRIP. In essence, an inelastic straining of the material results from stresses that are below the yield stress, and would not cause plastic flow in a classical plasticity sense. In this model, the TRIP effect is included in each phase transformation. Two parameters are required to describe the effect: the parameter $K_{\mathrm{s} \to \mathrm{d}}^{\mathrm{TRIP}}$ and the saturation function Φ. For simplicity, and with no additional experimental support, both are used with their default values for every phase transformation in the model.

Phase Plasticity

It is possible to allow for plasticity in the individual phases. By default, the equivalent plastic strain of the individual phases follows that of the compound material. That is to say, the equivalent plastic strain of a given phase in the Austenite Decomposition interface is equal to the equivalent plastic strain of a Plasticity node under Solid Mechanics. This equivalence is established through the Phase Transformation Strain multiphysics coupling. For the vanishing austenite, this is a reasonable modeling assumption. However, for phases that appear gradually and devoid of prior plastic straining, this assumption is questionable. Following Ref. 1, this deficiency can be remedied by allowing for plastic recovery of the destination phase. This is done for every destination phase in the model.

Initial Strains from Heating and Austenitization

To account for the strains that follow from thermal expansion and austenitization of the (unknown) base phase composition, an initial strain is applied. The initial strain is given by

$$\varepsilon_0 = 5 \cdot 10^{-3} \cdot \begin{pmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{pmatrix}$$

Results and Discussion

When the billet is cooled, the austenite decomposes into a combination of ferrite, pearlite, bainite, and martensite. Because of the inhomogeneous rate of cooling, the resulting phase composition will differ throughout the billet. For example, the ends of the billet experience a higher rate of cooling than the mid section. This suggests that the ferritic, pearlitic, and bainitc phase transformations are reduced in favor of the transformation to martensite, as this transformation is controlled by the amount of undercooling beneath the start temperature M_s, see Figure 2 (left).

During cooling, the material in the billet undergoes straining. Thermal strains result from the change in temperature, and mechanical stresses cause transformation induced plasticity (TRIP). Stresses exceed the initial yield stress of the compound material. This can be seen in Figure 2 (middle), where the largest equivalent plastic strain is observed on the surface of the billet. The quenching simulation computes residual stresses. In Figure 2 (right), the axial stress is shown. Note that the stresses are compressive on the surface of the billet. This is usually beneficial from a fatigue standpoint.

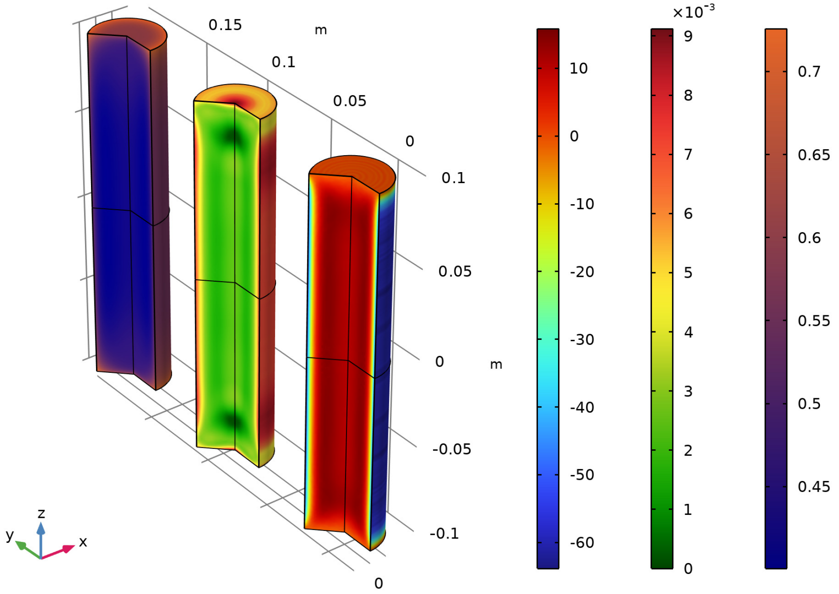

Figure 2: Phase fraction of martensite (left), equivalent plastic strain (middle), and axial tensile stress (right).

The evolving phase composition is shown for two locations at the mid plane of the billet: Figure 3 shows the phase composition on the surface, and Figure 4 the phase composition at the billet center. Comparing these phase compositions, the final fraction of martensite is higher at the surface than at the center of the billet. At the surface, the cooling rate is governed by the heat transfer coefficient and the temperature difference between the surface and the quenching oil. If the oil is able to provide a high enough rate of cooling, diffusion controlled phase transformations are limited in favor of the displacive martensitic transformation. In contrast, the cooling rate of a material point at the center of the billet is limited by the thermal diffusivity of the material. It is common to add certain alloying elements to the material to alter the phase transformation characteristics. This way certain diffusion-controlled phase transformations can be reduced or even suppressed.

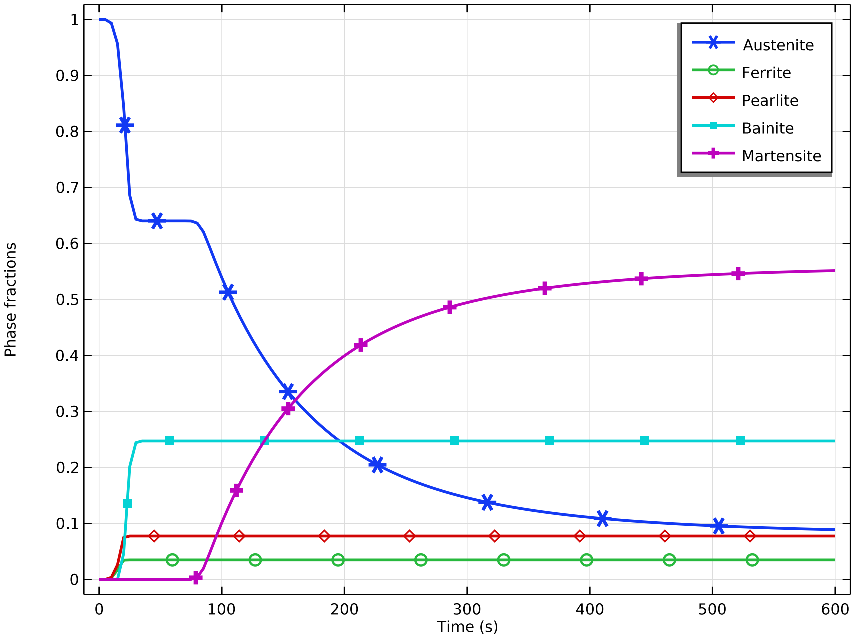

Figure 3: Phase composition on the surface of the steel billet middle.

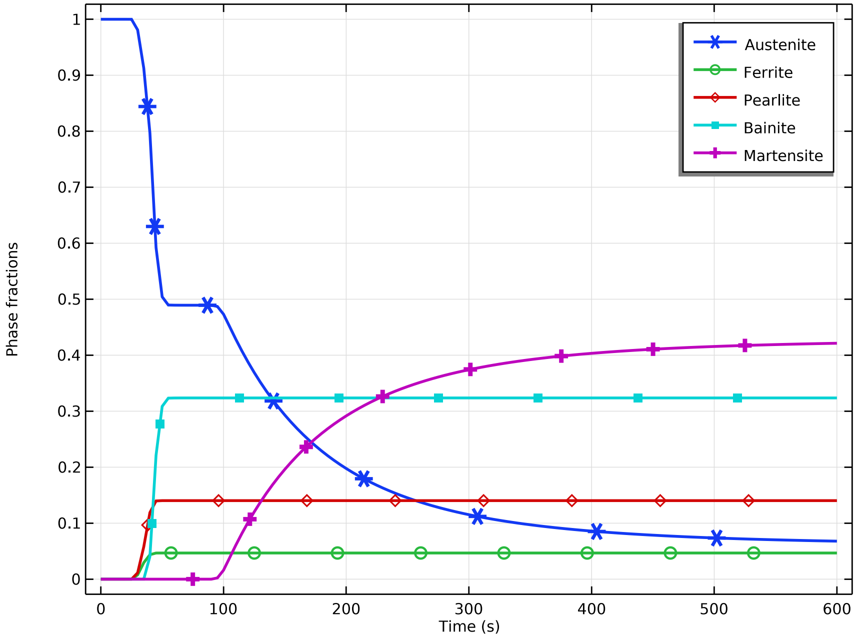

Figure 4: Phase composition at the billet center.

References

1. J.B. Leblond, “Mathematical modelling of transformation plasticity in steels II: Coupling with strain hardening phenomena,” *Int. J. Plast.*, vol. 5, pp. 573–591, 1989.

Application Library path: `Metal_Processing_Module/Steel_Quenching/quenching_of_a_steel_billet`

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** > **Steel Quenching**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **General Studies** > **Time Dependent**.
6. Click **Done**.

GEOMETRY 1

Create a node group that contains the temperature dependent data for the transformation of austenite to ferrite.

GLOBAL DEFINITIONS

Austenite to Ferrite

1. In the **Model Builder** window, right-click **Global Definitions** and choose **Node Group**.
2. In the **Settings** window for **Group**, type `Austenite to Ferrite` in the **Label** text field.

Interpolation 1 (int1)

1. In the **Home** toolbar, click **Functions** and choose **Global** > **Interpolation**.
2. In the **Settings** window for **Interpolation**, locate the **Definition** section.
3. In the **Function name** text field, type `K_Austenite_to_Ferrite`.
4. Click **Load from File**.
5. Browse to the model's Application Libraries folder and double-click the file `quenching_of_a_steel_billet_K_Austenite_to_Ferrite.txt`.
6. Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Piecewise cubic**.
7. Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
t	degC

8. In the **Function** table, enter the following settings:

Function	Unit
K_Austenite_to_Ferrite	1/s

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 `L_Austenite_to_Ferrite`.
4. Click **Load from File**.
5. Browse to the model's Application Libraries folder and double-click the file `quenching_of_a_steel_billet_L_Austenite_to_Ferrite.txt`.
6. Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Piecewise cubic**.
7. Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
t	degC

8. In the **Function** table, enter the following settings:

Function	Unit
L_Austenite_to_Ferrite	1/s

Create node groups and interpolation functions in a similar fashion for the austenite transformation into pearlite and bainite. Load the appropriate functions from file.

Read temperature dependent data for the Young's modulus.

Interpolation 7 (int7)

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 `EYoung`.
4. Click **Load from File**.
5. Browse to the model's Application Libraries folder and double-click the file `quenching_of_a_steel_billet_EYoung.txt`.
6. Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
t	degC

7 In the **Function** table, enter the following settings:

Function	**Unit**
EYoung	`GPa`

Read temperature dependent data for the heat transfer coefficient of the quenching oil.

Interpolation 8 (int8)

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 Click **Load from File**.

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

6 Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	**Unit**
t	`degC`

7 In the **Function** table, enter the following settings:

Function	**Unit**
htc	`W/(m^2*K)`

Create the geometry for the billet.

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 `0.02`.

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

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

AUSTENITE DECOMPOSITION (AUDC)

Ignore phase transformation latent heat in the model, but include traditional plasticity and TRIP.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Austenite Decomposition (audc)**.
2. In the **Settings** window for **Austenite Decomposition**, locate the **Heat Transfer** section.
3. Clear the **Enable phase transformation latent heat** checkbox.
4. Locate the **Solid Mechanics** section. Select the **Enable phase plasticity** checkbox.

   The volume reference temperatures for the phases are taken to be equal.

   Enter the material data for austenite. First, create the required phase materials.
5. Locate the **Material Properties** section. Click **Create Compound Material** in the upper-right corner of the section.

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 **Phase Material** section.
3. Click **Create Phase Material** in the upper-right corner of the section.
4. Locate the **Mechanical Properties** section. From the **Isotropic hardening model** list, choose **Linear**.

Ferrite

1. In the **Model Builder** window, click **Ferrite**.
2. In the **Settings** window for **Metallurgical Phase**, locate the **Phase Material** section.
3. Click **Create Phase Material** in the upper-right corner of the section.
4. Locate the **Mechanical Properties** section. From the **Isotropic hardening model** list, choose **Linear**.

Pearlite

1. In the **Model Builder** window, click **Pearlite**.
2. In the **Settings** window for **Metallurgical Phase**, locate the **Phase Material** section.
3. Click **Create Phase Material** in the upper-right corner of the section.
4. Locate the **Mechanical Properties** section. From the **Isotropic hardening model** list, choose **Linear**.

Bainite

1. In the **Model Builder** window, click **Bainite**.
2. In the **Settings** window for **Metallurgical Phase**, locate the **Phase Material** section.
3. Click **Create Phase Material** in the upper-right corner of the section.

4. Locate the **Mechanical Properties** section. From the **Isotropic hardening model** list, choose **Linear**.

Martensite

1. In the **Model Builder** window, click **Martensite**.
2. In the **Settings** window for **Metallurgical Phase**, locate the **Phase Material** section.
3. Click **Create Phase Material** in the upper-right corner of the section.
4. Locate the **Mechanical Properties** section. From the **Isotropic hardening model** list, choose **Linear**.

GLOBAL DEFINITIONS

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

Austenite (mat2)

In the **Model Builder** window, expand the **Global Definitions** > **Materials** node.

Interpolation 1 (int1)

1. In the **Model Builder** window, expand the **Austenite (mat2)** node.
2. Right-click **Global Definitions** > **Materials** > **Austenite (mat2)** > **Basic (def)** and choose **Functions** > **Interpolation**.
3. In the **Settings** window for **Interpolation**, locate the **Definition** section.
4. In the **Function name** text field, type `k`.
5. Click 📂 **Load from File**.
6. Browse to the model's Application Libraries folder and double-click the file `quenching_of_a_steel_billet_kAustenite.txt`.
7. Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
`t`	`degC`

8. In the **Function** table, enter the following settings:

Function	Unit
k	`W/(m*K)`

Austenite (mat2)

1. In the **Model Builder** window, under **Global Definitions** > **Materials** > **Austenite (mat2)** click **Basic (def)**.

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

3 Click + **Select Quantity**.

4 In the **Physical Quantity** dialog, type `temperature` in the text field.

5 In the tree, select **General** > **Temperature (K)**.

6 Click **OK**.

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 `Cp`.

4 Click **Load from File**.

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

6 Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	**Unit**
`t`	`degC`

7 In the **Function** table, enter the following settings:

Function	**Unit**
Cp	`J/(kg*K)`

Austenite (mat2)

1 In the **Model Builder** window, under **Global Definitions** > **Materials** > **Austenite (mat2)** click **Basic (def)**.

2 In the **Settings** window for **Basic**, locate the **Output Properties** section.

3 In the table, enter the following settings:

Property	**Variable**	**Expression**
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`k(T)`
Density	rho	`7930`
Heat capacity at constant pressure	Cp	`Cp(T)`

4 In the **Model Builder** window, under **Global Definitions** > **Materials** > **Austenite (mat2)** click **Thermal expansion (ThermalExpansion)**.

5 In the **Settings** window for **Thermal Expansion**, locate the **Output Properties** section.

6. In the table, enter the following settings:

Property	**Variable**	**Expression**
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	`2.2e-5`

7. In the **Model Builder** window, under **Global Definitions** > **Materials** > **Austenite (mat2)** click **Young's modulus and Poisson's ratio (Enu)**.
8. In the **Settings** window for **Young's Modulus and Poisson's Ratio**, locate the **Model Inputs** section.
9. Click + **Select Quantity**.
10. In the **Physical Quantity** dialog, select **General** > **Temperature (K)** in the tree.
11. Click **OK**.
12. In the **Settings** window for **Young's Modulus and Poisson's Ratio**, locate the **Output Properties** section.
13. In the table, enter the following settings:

Property	**Variable**	**Expression**
Young's modulus	E	`EYoung(T)`
Poisson's ratio	nu	`0.3`

14. In the **Model Builder** window, under **Global Definitions** > **Materials** > **Austenite (mat2)** click **Elastoplastic material model (ElastoplasticModel)**.
15. In the **Settings** window for **Elastoplastic Material Model**, locate the **Model Inputs** section.
16. Click + **Select Quantity**.
17. In the **Physical Quantity** dialog, select **General** > **Temperature (K)** in the tree.
18. Click **OK**.

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 `sY`.
4. Click **Load from File**.
5. Browse to the model's Application Libraries folder and double-click the file `quenching_of_a_steel_billet_sYAustenite.txt`.

6 Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
t	degC

7 In the **Function** table, enter the following settings:

Function	Unit
sY	MPa

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 `h`.

4 Click **Load from File**.

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

6 Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
t	degC

7 In the **Function** table, enter the following settings:

Function	Unit
h	GPa

Austenite (mat2)

1 In the **Model Builder** window, under **Global Definitions** > **Materials** > **Austenite (mat2)** click **Elastoplastic material model (ElastoplasticModel)**.

2 In the **Settings** window for **Elastoplastic Material Model**, locate the **Output Properties** section.

3 In the table, enter the following settings:

Property	Variable	Expression
Initial yield stress	sigmags	sY(T)
Isotropic tangent modulus	Et	h(T)

Enter the material data in a similar fashion for ferrite, pearlite, bainite, and martensite.

HEAT TRANSFER IN SOLIDS (HT)

Initial Values 1

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Heat Transfer in Solids (ht)** click **Initial Values 1**.
2. In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
3. In the T text field, type `900[degC]`.

Symmetry 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2. Select Boundary 2 only.

Heat Flux 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2. Select Boundaries 3 and 4 only.
3. In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
4. From the **Flux type** list, choose **Convective heat flux**.
5. In the h text field, type `htc(T)`.
6. In the T_{ext} text field, type `80[degC]`.

SOLID MECHANICS (SOLID)

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**.

   Use the hardening behavior of the compound material.
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**.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Initial Stress and Strain 1

1. In the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.
2. In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.

3 Specify the ε_0 matrix as

`0.005`	`0`	`0`
`0`	`0.005`	`0`
`0`	`0`	`0.005`

Symmetry Plane 1

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

2 Select Boundary 2 only.

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 **Model Input** section.

3 Click **Create Model Input** for **Volume reference temperature**.

SHARED PROPERTIES

Model Input 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Definitions** > **Shared Properties** click **Model Input 1**.

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

3 In the text field, type `900[degC]`.

AUSTENITE DECOMPOSITION (AUDC)

Austenite to Ferrite

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

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

3 In the $K_{\mathrm{s} \to \mathrm{d}}$ text field, type `K_Austenite_to_Ferrite(audc.T)`.

4 In the $L_{\mathrm{s} \to \mathrm{d}}$ text field, type `L_Austenite_to_Ferrite(audc.T)`.

5 Locate the **Phase Transformation Strain** section. Select the **Transformation-induced plasticity** checkbox.

6 Select the **Plastic recovery for destination phase** checkbox.

Austenite to Pearlite

1. In the **Model Builder** window, click **Austenite to Pearlite**.
2. In the **Settings** window for **Phase Transformation**, locate the **Phase Transformation** section.
3. In the $K_{\mathrm{s -> d}}$ text field, type `K_Austenite_to_Pearlite(audc.T)`.
4. In the $L_{\mathrm{s -> d}}$ text field, type `L_Austenite_to_Pearlite(audc.T)`.
5. Locate the **Phase Transformation Strain** section. Select the **Transformation-induced plasticity** checkbox.
6. Select the **Plastic recovery for destination phase** checkbox.

Austenite to Bainite

1. In the **Model Builder** window, click **Austenite to Bainite**.
2. In the **Settings** window for **Phase Transformation**, locate the **Phase Transformation** section.
3. In the $K_{\mathrm{s -> d}}$ text field, type `K_Austenite_to_Bainite(audc.T)`.
4. In the $L_{\mathrm{s -> d}}$ text field, type `L_Austenite_to_Bainite(audc.T)`.
5. Locate the **Phase Transformation Strain** section. Select the **Transformation-induced plasticity** checkbox.
6. Select the **Plastic recovery for destination phase** checkbox.

Austenite to Martensite

1. In the **Model Builder** window, click **Austenite to Martensite**.
2. In the **Settings** window for **Phase Transformation**, locate the **Phase Transformation** section.
3. In the M_{s} text field, type `300[degC]`.
4. Locate the **Phase Transformation Strain** section. Select the **Transformation-induced plasticity** checkbox.
5. Select the **Plastic recovery for destination phase** checkbox.

MESH 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Sequence Type** section.
3. From the list, choose **User-controlled mesh**.

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 From the **Predefined** list, choose **Extra fine**.

Boundary Layers 1

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

Boundary Layer Properties

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

2 Select Boundaries 3 and 4 only.

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

4 In the **Number of layers** text field, type `6`.

5 In the **Stretching factor** text field, type `1.5`.

6 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

Step 1: Time Dependent

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

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

3 In the **Output times** text field, type `range(0,5,600)`.

4 Right-click **Study 1** > **Step 1: Time Dependent** and choose **Get Initial Value for Step**.

5 Right-click **Step 1: Time Dependent** and choose **Get Initial Value for Step**.

STUDY 1

Solver Configurations

In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** node.

Solution 1 (sol1)

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

2 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

3 From the **Steps taken by solver** list, choose **Intermediate**.

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

Set preferred units for result presentation.

RESULTS

Preferred Units 1

1. In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.
2. In the **Settings** window for **Preferred Units**, locate the **Units** section.
3. Click **Add Physical Quantity**.
4. In the **Physical Quantity** dialog, select **General** > **Temperature (K)** in the tree.
5. Click **OK**.
6. In the **Settings** window for **Preferred Units**, locate the **Units** section.
7. In the table, enter the following settings:

Quantity	Unit	Preferred unit
Temperature	K	°C

8. Click **Add Physical Quantity**.
9. In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.
10. Click **OK**.
11. In the **Settings** window for **Preferred Units**, locate the **Units** section.
12. In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	MPa

13. Click **Apply**.

Phase fractions at the billet center

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Phase fractions at the billet center` in the **Label** text field.
3. Click to expand the **Title** section. From the **Title type** list, choose **None**.
4. Locate the **Plot Settings** section.
5. Select the **y-axis label** checkbox. In the associated text field, type `Phase fractions`.

Point Graph 1

1. Right-click **Phase fractions at the billet center** and choose **Point Graph**.
2. Select Point 1 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Austenite Decomposition** > **Austenite** > **audc.phase1.xiGp - Phase fraction - 1**.

4 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

6 From the **Positioning** list, choose **Interpolated**.

7 Click to expand the **Legends** section. Select the **Show legends** checkbox.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`Austenite`

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `audc.phase2.xiGp`.

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

Legends
`Ferrite`

Point Graph 3

1 In the **Model Builder** window, under **Results** > **Phase fractions at the billet center** right-click **Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `audc.phase3.xiGp`.

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

Legends
`Pearlite`

Point Graph 4

1 Right-click **Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `audc.phase4.xiGp`.

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

Legends
`Bainite`

Point Graph 5

1 Right-click **Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `audc.phase5.xiGp`.

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

Legends
`Martensite`

Phase fractions at the billet surface

1 In the **Model Builder** window, right-click **Phase fractions at the billet center** and choose **Duplicate**.

2 In the **Model Builder** window, click **Phase fractions at the billet center 1**.

3 In the **Settings** window for **1D Plot Group**, type `Phase fractions at the billet surface` in the **Label** text field.

Point Graph 1

1 In the **Model Builder** window, click **Point Graph 1**.

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

3 Click to select the **Activate Selection** toggle button.

4 Select Point 3 only.

Point Graph 2

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

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

3 Click to select the **Activate Selection** toggle button.

4 Select Point 3 only.

Point Graph 3

1 In the **Model Builder** window, click **Point Graph 3**.

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

3 Click to select the **Activate Selection** toggle button.

4 Select Point 3 only.

Point Graph 4

1. In the **Model Builder** window, click **Point Graph 4**.
2. In the **Settings** window for **Point Graph**, locate the **Selection** section.
3. Click to select the **Activate Selection** toggle button.
4. Select Point 3 only.

Point Graph 5

1. In the **Model Builder** window, click **Point Graph 5**.
2. In the **Settings** window for **Point Graph**, locate the **Selection** section.
3. Click to select the **Activate Selection** toggle button.
4. Select Point 3 only.
5. In the **Phase fractions at the billet surface** toolbar, click **Plot**.

Axial stress profile

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Axial stress profile` in the **Label** text field.
3. Locate the **Data** section. From the **Time selection** list, choose **Last**.

Line Graph 1

1. Right-click **Axial stress profile** and choose **Line Graph**.
2. Select Boundary 2 only.
3. In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Stress** > **Stress tensor (spatial frame) - N/m^2** > **solid.sGpzz - Stress tensor, zz-component**.
4. Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.
5. Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
6. In the **Expression** text field, type `R`.
7. Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.
8. Click to expand the **Quality** section. In the **Axial stress profile** toolbar, click **Plot**.

Mirror 3D 1

1. In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.
2. In the **Settings** window for **Mirror 3D**, locate the **Plane Data** section.
3. From the **Plane** list, choose **XY-planes**.

3D Plot Group 17

1 In the **Results** toolbar, click **3D Plot Group**.

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

3 From the **Dataset** list, choose **Mirror 3D 1**.

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

5 Click to expand the **Plot Array** section. Select the **Enable** checkbox.

6 From the **Array axis** list, choose **y**.

7 From the **Padding** list, choose **Absolute**.

8 In the **Padding length** text field, type `0.05`.

Axial stress

1 Right-click **3D Plot Group 17** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Axial stress` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Stress** > **Stress tensor (spatial frame) - N/m^2** > **solid.sGpzz - Stress tensor, zz-component**.

4 Locate the **Expression** section. From the **Unit** list, choose **MPa**.

Equivalent plastic strain

1 Right-click **Axial stress** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Equivalent plastic strain` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Strain** > **solid.epeGp - Equivalent plastic strain - 1**.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Traffic**.

Surface 1

1 In the **Model Builder** window, expand the **Results** > **Martensite, 3D (audc)** node.

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

Martensite phase fraction

1 In the **Model Builder** window, right-click **3D Plot Group 17** and choose **Paste Surface**.

2 In the **Settings** window for **Surface**, type `Martensite phase fraction` in the **Label** text field.