Created in COMSOL Multiphysics 5.3

Friction Stir Welding of an Aluminum Plate

Introduction

Manufacturers use a modern welding method called friction stir welding to join aluminum plates. This application analyzes the heat transfer in this welding process. The model is based on a paper by M. Song and R. Kovacevic (Ref. 1).

In friction stir welding, a rotating tool moves along the weld joint and softens the aluminum through the generation of friction heat. The tool's rotation stirs the soften aluminum such that the two plates are joined. Figure 1 shows the rotating tool and the aluminum plates being are joined.

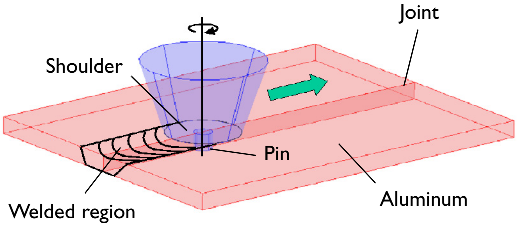


Figure 1: Two aluminum plates being joined by friction stir welding.

The rotating tool is in contact with the aluminum plates along two surfaces: the tool's *shoulder*, and the tool's *pin*. The tool adds heat to the aluminum plates through both interfaces.

During the welding process, the tool moves along the weld joint. This movement would require a fairly complex model if you want to model the tool as a moving heat source. This example takes a different approach that uses a moving coordinate system that is fixed at the tool axis (Ref. 1 also takes this approach). After making the coordinate transformation, the heat transfer problem becomes a stationary convection-conduction problem that is straightforward to model.

The model includes some simplifications. For example, the coordinate transformation assumes that the aluminum plates are infinitely long. This means that the analysis neglects effects near the edges of the plates. Neither does the model account for the stirring process in the aluminum, which is very complex because it includes phase changes and material flow from the front to the back of the rotating tool.

Model Definition

The model geometry is symmetric around the weld. It is therefore sufficient to model only one aluminum plate. The plate dimensions are 120×102×12.7 mm, surrounded by two infinite domains in the x-direction. Figure 2 shows the resulting model geometry:

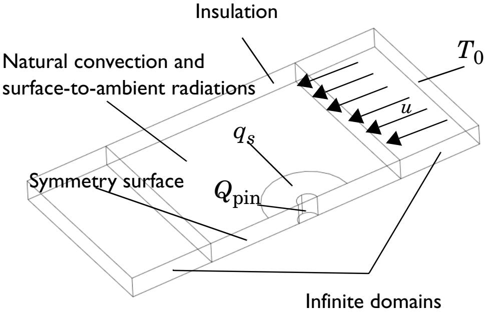


Figure 2: Model geometry for friction stir welding.

The following equation describes heat transfer in the plate. As a result of fixing the coordinate system in the welding tool, the equation includes a convective term in addition to the conductive term. The equation is

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

where k represents thermal conductivity, ρ is the density, C_p denotes specific heat capacity, and $\mathbf{u}$ is the velocity.

The model sets the velocity to $1.59 \cdot 10^{-3}$ m/s in the negative x direction.

The model simulates the heat generated in the interface between the tool's pin and the workpiece as a surface heat source (expression adapted from Ref. 2):

$$q_{\text{pin}}(T) = \frac{\mu}{\sqrt{3(1+\mu^2)}} r_{\text{p}} \omega \overline{Y}(T)$$

Here μ is the friction coefficient, r_{p} denotes the pin radius, ω refers to the pin's angular velocity (rad/s), and $\overline{Y}(T)$ is the average shear stress of the material. As indicated, the average shear stress is a function of the temperature; for this tutorial, you approximate this function with an interpolation function determined from experimental data given in Ref. 1

(see Figure 3).

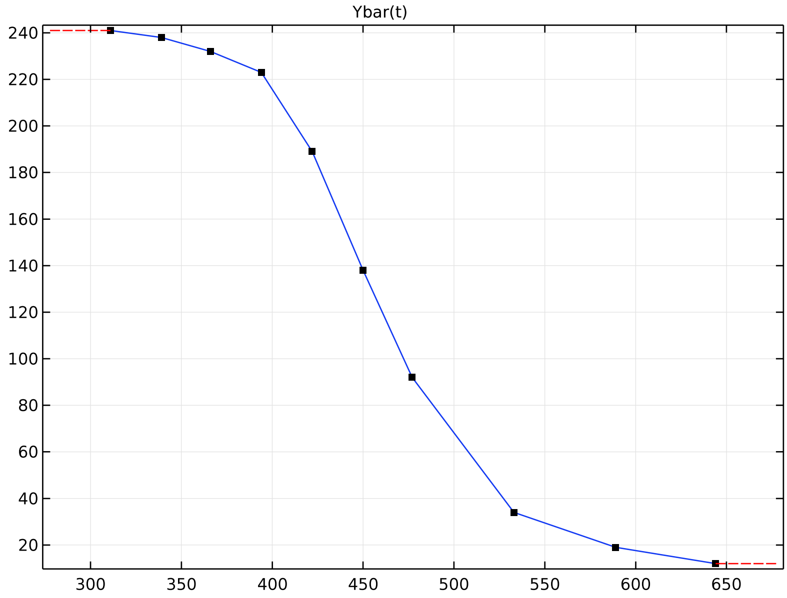


Figure 3: Yield stress (MPa) vs. temperature (K) for 6061-T6 aluminum.

Additionally, heat is generated at the interface between the tool's shoulder and the workpiece; the following expression defines the local heat flux per unit area ($\mathrm{W/m^2}$) at the distance r from the center axis of the tool:

$$q_{\mathrm{shoulder}}(r, T) = \begin{cases} (\mu F_{\mathrm{n}}/A_{\mathrm{s}})\omega r & \text{if } T < T_{\mathrm{melt}} \\ 0 & \text{if } T \geq T_{\mathrm{melt}} \end{cases}$$

Here F_{n} represents the normal force, A_{s} is the shoulder's surface area, and T_{melt} is aluminum's melting temperature. As before, μ is the friction coefficient and ω is the angular velocity of the tool (rad/s).

Above the melting temperature of aluminum, the friction between the tool and the aluminum plate is very low. Therefore, the model sets the heat generation from the shoulder and the pin to zero when the temperature is equal to or higher than the melting temperature.

Symmetry is assumed along the weld joint boundary.

The upper and lower surfaces of the aluminum plates lose heat due to natural convection and surface-to-ambient radiation. The corresponding heat flux expressions for these surfaces are

$$\begin{aligned} q_\mathrm{u} &= h_\mathrm{u}(T_0 - T) + \varepsilon\sigma(T_\mathrm{amb}^4 - T^4) \\ q_\mathrm{d} &= h_\mathrm{d}(T_0 - T) + \varepsilon\sigma(T_\mathrm{amb}^4 - T^4) \end{aligned}$$

where h_u and h_d are heat transfer coefficients for natural convection, T_0 is an associated reference temperature, ε is the surface emissivity, σ is the Stefan-Boltzmann constant, and T_amb is the ambient air temperature.

The modeling of an infinite domain on the left-hand side, where the aluminum leaves the computational domain, makes sure that the temperature is in equilibrium with the temperature at infinity through natural convection and surface-to-ambient radiation. You therefore set the boundary condition to insulation at that location.

You can compute values for the heat transfer coefficients using empirical expressions available in the heat-transfer literature, for example, Ref. 3. In this application, use the values $h_\mathrm{u} = 12.25$ W/(m^2·K) and $h_\mathrm{d} = 6.25$ W/(m^2·K)

Results and Discussion

Figure 4 shows the resulting temperature field. Consider this result as what you would see through a window fixed to the moving welding tool.

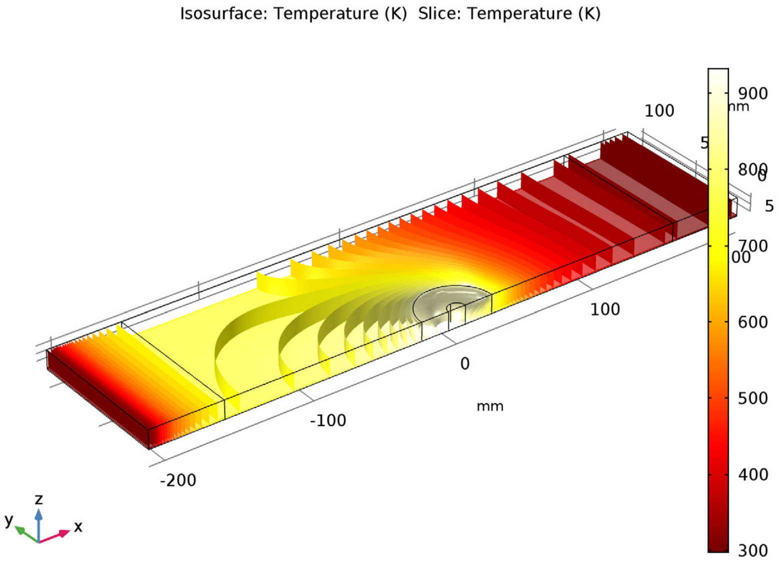


Figure 4: Temperature field in the aluminum plate.

The temperature is highest where the aluminum is in contact with the rotating tool. Behind the tool, the process transports hot material away, while in front of the tool, new cold material enters.

References

1. M. Song and R. Kovacevic, “Thermal modeling of friction stir welding in a moving coordinate system and its validation,” *Int’l J. of Machine Tools & Manufacture*, vol. 43, pp. 605–615, 2003.

2. P. Colegrove et al., “3-dimensional Flow and Thermal Modelling of the Friction Stir Welding Process,” *Proceedings of the 2nd International Symposium on Friction Stir Welding*, Gothenburg, Sweden, 2000.

3. A. Bejan, *Heat Transfer*, John Wiley & Sons, 1993.

Application Library path: `Heat_Transfer_Module/Thermal_Contact_and_Friction/friction_stir_welding`

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 **3D**.
2. In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Stationary**.
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters

1. On the **Home** toolbar, click **Parameters**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.
3. In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
`T_melt`	`933[K]`	933 K	`Workpiece melting temperature`
`h_upside`	`12.25[W/(m^2*K)]`	12.25 W/(m²·K)	`Heat transfer coefficient, upside`
`h_downside`	`6.25[W/(m^2*K)]`	6.25 W/(m²·K)	`Heat transfer coefficient, downside`
`epsilon`	`0.3[1]`	0.3	`Surface emissivity`
`u_weld`	`1.59[mm/s]`	0.00159 m/s	`Welding speed`

Name	**Expression**	**Value**	**Description**
mu	0.4[1]	0.4	Friction coefficient
n	637[1/min]	10.617 1/s	Rotation speed (RPM)
omega	2*pi[rad]*n	66.706 rad/s	Angular velocity (rad/s)
F_n	25[kN]	25000 N	Normal force
r_pin	6[mm]	0.006 m	Pin radius
r_shoulder	25[mm]	0.025 m	Shoulder radius
A_s	pi*(r_shoulder^2-r_pin^2)	0.0018504 m^2	Shoulder surface area

Interpolation 1 (int1)

1 On 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 `Ybar`.

4 In the table, enter the following settings:

t	**f(t)**
311	241
339	238
366	232
394	223
422	189
450	138
477	92
533	34
589	19
644	12

5 Click **Plot**.

If you have entered the numbers correctly, the curve should look like that in Figure 3.

Step 1 (step1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Step**.

2 In the **Settings** window for **Step**, click to expand the **Smoothing** section.

3 In the **Size of transition zone** text field, type `5`.

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**.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

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

3 In the **Width** text field, type `320`.

4 In the **Depth** text field, type `102`.

5 In the **Height** text field, type `12.7`.

6 Locate the **Position** section. In the **x** text field, type `-160`.

7 Click **Build Selected**.

Block 2 (blk2)

1 On the **Geometry** toolbar, click **Block**.

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

3 In the **Width** text field, type `420`.

4 In the **Depth** text field, type `102`.

5 In the **Height** text field, type `12.7`.

6 Locate the **Position** section. In the **x** text field, type `-210`.

7 Click **Build Selected**.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `r_shoulder`.

4 In the **Height** text field, type `12.7`.

5 Click **Build Selected**.

Cylinder 2 (cyl2)

1 On the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `r_pin`.

4 In the **Height** text field, type `12.7`.

5 Click **Build Selected**.

Block 3 (blk3)

1 On the **Geometry** toolbar, click **Block**.

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

3 In the **Width** text field, type `2*r_shoulder`.

4 In the **Depth** text field, type `r_shoulder`.

5 In the **Height** text field, type `12.7`.

6 Locate the **Position** section. In the **x** text field, type `-r_shoulder`.

7 In the **y** text field, type `-r_shoulder`.

8 Click **Build Selected**.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the objects **cyl1** and **cyl2** only.

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

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **blk3** only.

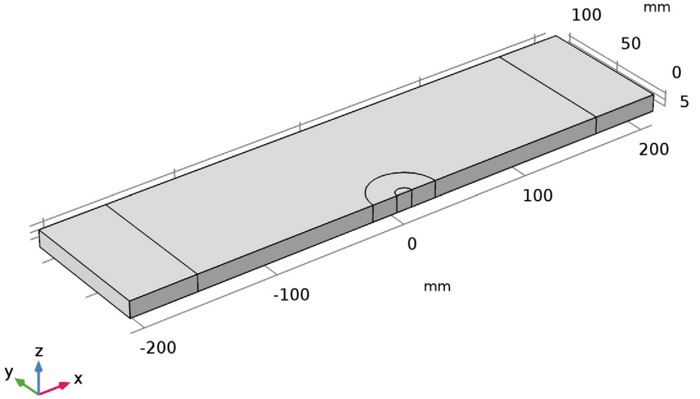
Form Union (fin)

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**, click **Build Selected**.

The model geometry is now complete.

3 Click the **Zoom Extents** button on the **Graphics** toolbar to see the entire geometry.



DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

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

4 Select Boundary 14 only.

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

Name	**Expression**	**Unit**	**Description**
`R`	`sqrt(x^2+y^2)`	m	`Distance in xy-plane from tool center axis`
`q_shoulder`	`(mu*F_n/A_s)*(R*omega)*step1((T_melt-T)[1/K])`	W/m²	`Surface heat source, shoulder-workpiece interface`

Variables 2

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

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

4 Select Boundaries 15 and 19 only.

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

Name	Expression	Unit	Description
`q_pin`	`mu/sqrt(3*(1+mu^2))*(r_pin*omega)*Ybar(T[1/K])[MPa]*step1((T_melt-T)[1/K])`	W/m²	`Surface heat source, pin-workpiece interface`

HEAT TRANSFER IN SOLIDS (HT)

Set the ambient temperature to be used in boundary conditions and initial values of the Heat Transfer interface.

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 **Ambient Settings** section.
3. In the T_{amb} text field, type `300[K]`.

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**, choose **Ambient temperature (ht)** from the T list.

Solid 1

The domain selection for the default equation model is fixed to all domains to ensure that no domain lacks a defining equation. To modify the equation model for some specific domains, you simply add nodes that override the default equation.

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

Translational Motion 1

1. On the **Physics** toolbar, click **Attributes** and choose **Translational Motion**.
2. In the **Settings** window for **Translational Motion**, locate the **Translational Motion** section.
3. Specify the $\mathbf{u}_{\mathrm{trans}}$ vector as

`-u_weld`	x
`0`	y
`0`	z

DEFINITIONS

Infinite Element Domain 1 (ie1)

1. On the **Definitions** toolbar, click **Infinite Element Domain**.
2. Select Domains 1 and 5 only.

HEAT TRANSFER IN SOLIDS (HT)

Diffuse Surface 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.
2. Select Boundaries 3, 4, 8, 9, 13, 25, and 26 only.

   Together, these boundaries form the top and bottom surfaces of the geometry.
3. In the **Settings** window for **Diffuse Surface**, locate the **Surface Emissivity** section.
4. From the ε list, choose **User defined**. In the associated text field, type `epsilon`.
5. Locate the **Ambient** section. From the T_{amb} list, choose **Ambient temperature (ht)**.

Outflow 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.
2. Select Boundary 1 only.

Heat Flux 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2. Select Boundaries 3, 8, 13, and 25 only.
3. In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
4. Click the **Convective heat flux** button.
5. In the h text field, type `h_downside`.
6. From the T_{ext} list, choose **Ambient temperature (ht)**.

Heat Flux 2

1. On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2. Select Boundaries 4, 9, and 26 only.
3. In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
4. Click the **Convective heat flux** button.
5. In the h text field, type `h_upside`.
6. From the T_{ext} list, choose **Ambient temperature (ht)**.

Heat Flux 3

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

2 Select Boundary 14 only.

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

4 In the q_0 text field, type `q_shoulder`.

Boundary Heat Source 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Heat Source**.

2 Select Boundaries 15 and 19 only.

3 In the **Settings** window for **Boundary Heat Source**, locate the **Boundary Heat Source** section.

4 In the Q_b text field, type `q_pin`.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 28 only.

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

4 From the T_0 list, choose **Ambient temperature (ht)**.

MATERIALS

Now specify the materials. By default, the first material you add applies to all domains. To specify a different material in some domains you simply add another material for those domains.

ADD MATERIAL

1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-In>Aluminum**.

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

MATERIALS

Aluminum (mat1)

1 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Add a material for the pin and specify the required properties.

Material 2 (mat2)

1. On the **Materials** toolbar, click **Blank Material**.
2. In the **Settings** window for **Material**, type `Pin` in the **Label** text field.
3. Select Domain 4 only.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Thermal conductivity	k	`42[W/(m*K)]`	W/(m·K)	Basic
Density	rho	`7800[kg/m^3]`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`500[J/(kg*K)]`	J/(kg·K)	Basic

MESH 1

On the **Mesh** toolbar, click **Boundary** and choose **Free Quad**.

Free Quad 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Quad 1**.
2. Select Boundaries 4, 9, and 26 only.

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 **Extremely fine**.
4. Click **Build All**.
5. On the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

Free Triangular 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Triangular 1**.
2. Select Boundaries 14 and 18 only.
3. On the **Mesh** toolbar, click **Size Attribute** and choose **Normal**.

Size 1

Click **Swept**.

Swept 1

Click **Distribution**.

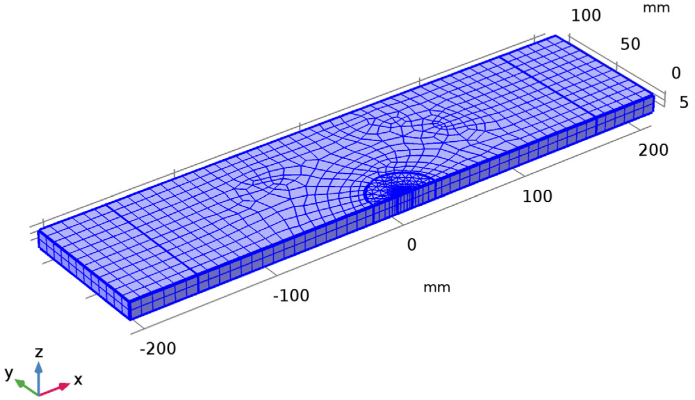
Distribution 1

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

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

3 In the **Number of elements** text field, type 2.

4 Click **Build All**.



STUDY 1

For this fairly small problem, use a direct solver for faster convergence.

Solution 1 (sol1)

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

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

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node.

4 Right-click **Direct** and choose **Enable**.

5 On the **Study** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The first default plot group shows the temperature field as a surface plot. Use the second default plot group as the starting point for reproducing the plot in Figure 4.

Isosurface

1 In the **Model Builder** window, expand the **Results>Isothermal Contours (ht)** node, then click **Isosurface**.

2 In the **Settings** window for **Isosurface**, locate the **Levels** section.

3 From the **Entry method** list, choose **Levels**.

4 In the **Levels** text field, type `range(300,20,980)`.

5 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

Isothermal Contours (ht)

1 In the **Model Builder** window, under **Results** click **Isothermal Contours (ht)**.

2 On the **Isothermal Contours (ht)** toolbar, click **Slice**.

Slice 1

1 In the **Model Builder** window, under **Results>Isothermal Contours (ht)** click **Slice 1**.

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

3 From the **Plane** list, choose **XY-planes**.

4 From the **Entry method** list, choose **Coordinates**.

5 In the **Z-coordinates** text field, type `1`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

7 On the **Isothermal Contours (ht)** toolbar, click **Plot**.