Model created in COMSOL Multiphysics 6.4

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 joined.

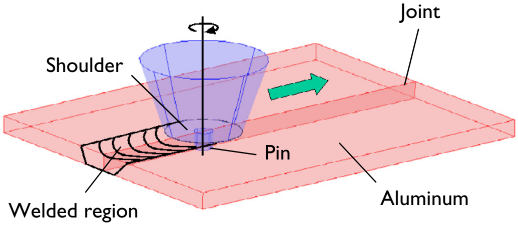


Figure 1: Two aluminum plates being joined by friction stir welding.

The rotating tool is in contact with the aluminum plates along two surfaces: the tool's *shoulder*, and the tool's *pin*. The tool heats 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 to 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-by-102-by-12.7 mm, surrounded by two infinite domains in the x direction. Figure 2 shows the resulting model geometry:

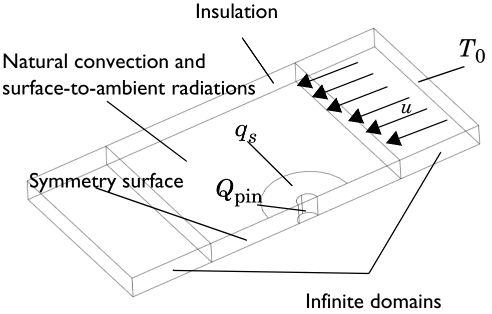


Figure 2: Model geometry for friction stir welding.

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 at the interface between the tool's pin and the workpiece as a surface heat source (expression adapted from Ref. 2):

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

Here μ is the friction coefficient, r_{p} denotes the pin radius, ω refers to the pin's angular velocity (rad/s), and $\bar{Y}(T)$ 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).

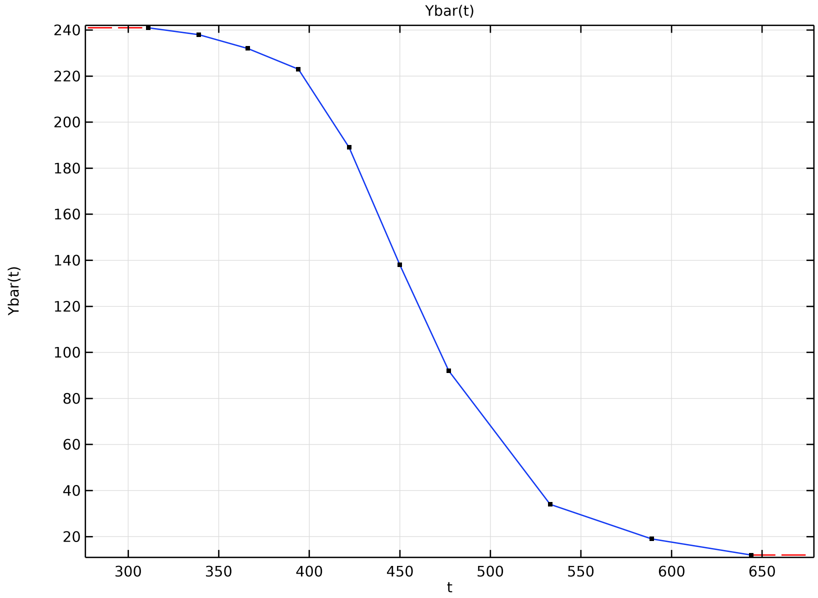


Figure 3: Yield stress (MPa) vs. temperature (K) for 6061-T6 aluminum.

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 surface area, and T_{melt} is the aluminum 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.

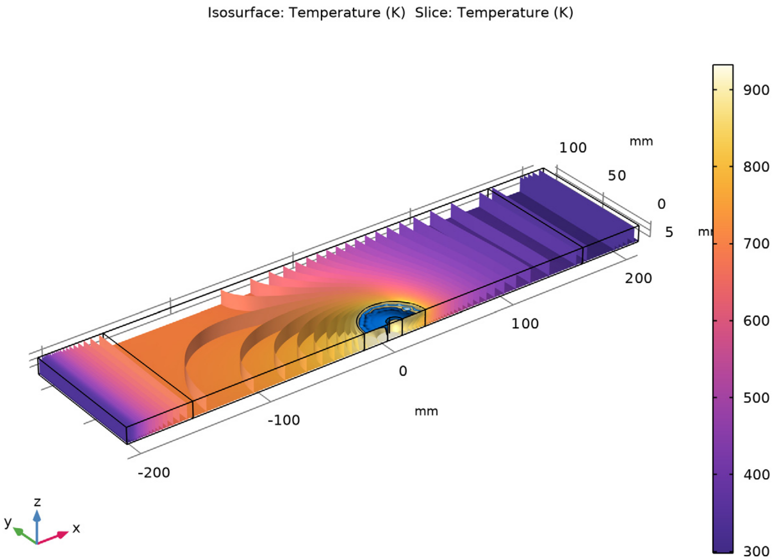


Figure 4: Temperature field in the aluminum plate.

The temperature is highest where the aluminum is in contact with the rotating tool. The blue area shows where the aluminum exceeds the melting temperature. Numerical simulations can be used as a predictive tool for calibration. Here the tool of the welding machine is rotating too fast. 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 and others, “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 **General Studies** > **Stationary**.
6. Click **Done**.

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
`T_melt`	`933[K]`	933 K	`Workpiece melting temperature`
`h_upside`	`12.25[W/(m^2*K)]`	12.25 W/(m^2·K)	`Heat transfer coefficient, upside`
`h_downside`	`6.25[W/(m^2*K)]`	6.25 W/(m^2·K)	`Heat transfer coefficient, downside`

Name	Expression	Value	Description
epsilon	0.3[1]	0.3	Surface emissivity
u_weld	1.59[mm/s]	0.00159 m/s	Welding speed
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 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 `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 In the **Home** toolbar, click f(x) **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 In 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 In 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 In 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 In 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)

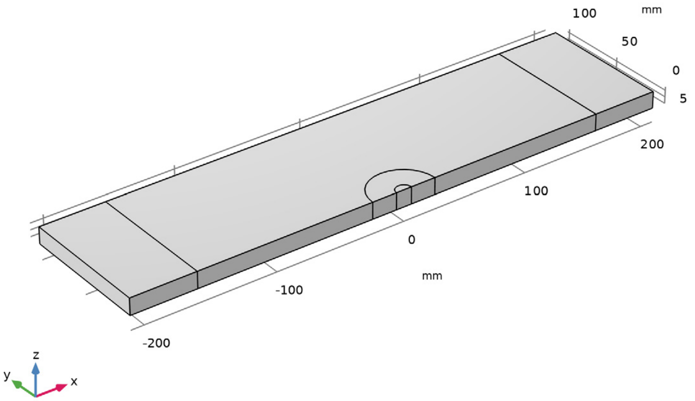
1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for **Block**, locate the **Size and Shape** section.
3. In the **Width** text field, type `2*r_shoulder`.
4. In the **Depth** text field, type `r_shoulder`.
5. In the **Height** text field, type `12.7`.
6. Locate the **Position** 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. In 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. Click to select the **Activate Selection** toggle button for **Objects to subtract**.
5. Select the object **blk3** only.
6. In the **Geometry** toolbar, click **Build All**.

   The model geometry is now complete.

7 Click the ⊹ **Zoom Extents** button in the **Graphics** toolbar to see the entire geometry.



DEFINITIONS

Variables 1

1 In the **Definitions** toolbar, click a= **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 In the **Definitions** toolbar, click a= **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/m^2	`Surface heat source, pin-workpiece interface`

Ambient Properties 1 (ampr1)

1. In the **Physics** toolbar, click **Shared Properties** and choose **Ambient Properties**.

   Set the ambient temperature to be used as boundary conditions and initial values of the **Heat Transfer** interface.
2. In the **Settings** window for **Ambient Properties**, locate the **Ambient Conditions** section.
3. In the T_{amb} text field, type `300[K]`.

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. From the T list, choose **Ambient temperature (ampr1)**.

Solid with Translational Motion 1

1. In the **Physics** toolbar, click **Domains** and choose **Solid with Translational Motion**.
2. In the **Settings** window for **Solid with Translational Motion**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **All domains**.

Translational Motion 1

1. In the **Model Builder** window, click **Translational Motion 1**.
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. In the **Definitions** toolbar, click **Infinite Element Domain**.
2. Select Domains 1 and 5 only.

HEAT TRANSFER IN SOLIDS (HT)

Surface-to-Ambient Radiation 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Surface-to-Ambient Radiation**.
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 **Surface-to-Ambient Radiation**, locate the **Surface-to-Ambient Radiation** section.
4. From the ε list, choose **User defined**. In the associated text field, type `epsilon`.
5. From the T_{amb} list, choose **Ambient temperature (ampr1)**.

Heat Flux 1

1. In 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. From the **Flux type** list, choose **Convective heat flux**.
5. In the h text field, type `h_downside`.
6. From the T_{ext} list, choose **Ambient temperature (ampr1)**.

Heat Flux 2

1. In 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. From the **Flux type** list, choose **Convective heat flux**.
5. In the h text field, type `h_upside`.
6. From the T_{ext} list, choose **Ambient temperature (ampr1)**.

Heat Flux 3

1. In 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. In 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. In 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 (ampr1)**.

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. 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** > **Aluminum**.
4. Click the **Add to Component** button in the window toolbar.
5. In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Aluminum (mat1)

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

Pin

1. In 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	**Variable**	**Value**	**Unit**	**Property group**
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`42[W/(m*K)]`	W/(m·K)	Basic
Density	rho	`7800[kg/m^3]`	kg/m^3	Basic
Heat capacity at constant pressure	Cp	`500[J/(kg*K)]`	J/(kg·K)	Basic

MESH 1

Free Quad 1

1. In the **Mesh** toolbar, click **More Generators** and choose **Free Quad**.
2. Select Boundaries 4, 9, and 26 only.

Size

1. In the **Model Builder** window, click **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Extremely fine**.

Free Triangular 1

1. In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.
2. Select Boundaries 14 and 18 only.

Size 1

In the **Mesh** toolbar, click **Size Attribute** and choose **Normal**.

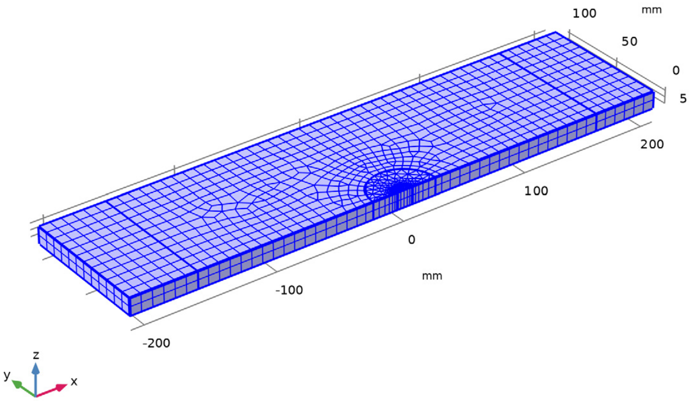
Swept 1

In the **Mesh** toolbar, click **Swept**.

Distribution 1

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

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

RESULTS

Temperature (ht)

The first default plot group shows the temperature field as a volume plot. Add a plot group from the **Result Templates** as the starting point for reproducing the plot in Figure 4.

RESULT TEMPLATES

1. In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.
2. Go to the **Result Templates** window.
3. In the tree, select **Study 1/Solution 1 (sol1)** > **Heat Transfer in Solids** > **Isothermal Contours (ht)**.
4. Click the **Add Result Template** button in the window toolbar.
5. In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Isosurface 1

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

Isothermal Contours (ht)

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

Slice 1

1. In the **Isothermal Contours (ht)** toolbar, click **Slice**.
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 **HeatCameraLight**.
7. In the **Isothermal Contours (ht)** toolbar, click **Plot**.

Now, add a volume plot that highlights the area where aluminum has melted.

Volume 1

1. Right-click **Isothermal Contours (ht)** and choose **Volume**.
2. In the **Settings** window for **Volume**, click to expand the **Title** section.
3. From the **Title type** list, choose **None**.
4. Locate the **Coloring and Style** section. From the **Color table** list, choose **JupiterAuroraBorealis**.
5. Clear the **Color legend** checkbox.

Selection 1

1. Right-click **Volume 1** and choose **Selection**.

   Select the parts of the geometry that belong to the aluminum plate.
2. Select Domains 1–3 and 5 only.

Filter 1

1. In the **Model Builder** window, right-click **Volume 1** and choose **Filter**.
2. In the **Settings** window for **Filter**, locate the **Element Selection** section.
3. In the **Logical expression for inclusion** text field, type `T>T_melt`.
4. In the **Isothermal Contours (ht)** toolbar, click **Plot**.