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.

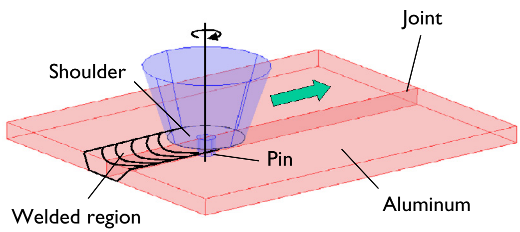


Figure 1: Two aluminum plates being joined by friction stir welding.

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-by-102-by-12.7 mm, surrounded by two infinite domains in the x direction. Figure 2 shows the resulting model geometry:

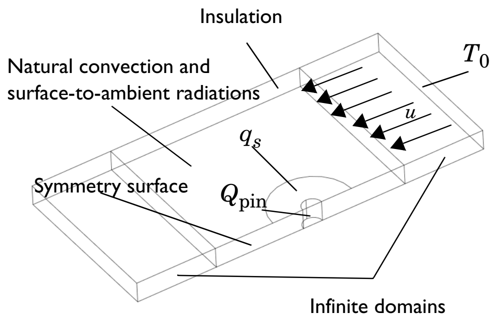


Figure 2: Model geometry for friction stir welding.

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

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

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).

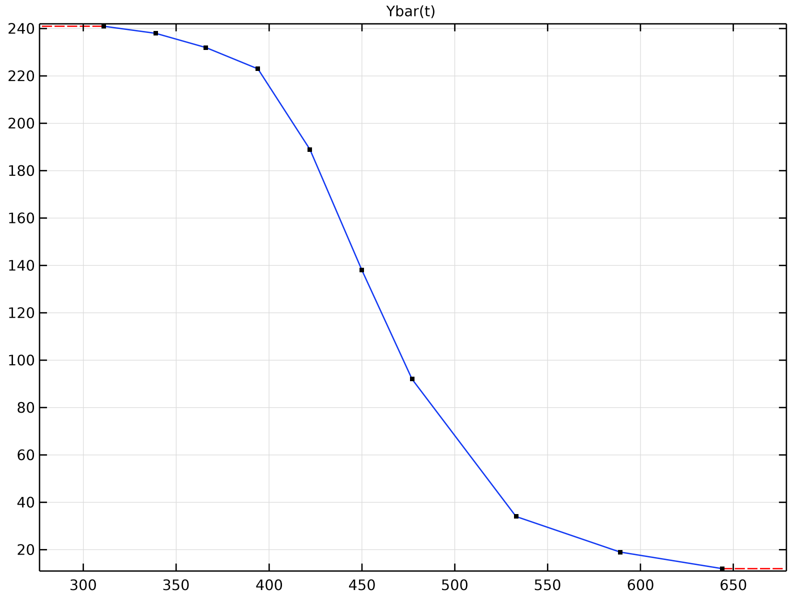


Figure 3: Yield stress (MPa) vs. temperature (K) for 6061-T6 aluminum.

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 (W/m^2) at the distance r from the center axis of the tool:

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

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

$$q_u = h_u(T_0 - T) + \varepsilon\sigma(T_{amb}^4 - T^4)$$
$$q_d = h_d(T_0 - T) + \varepsilon\sigma(T_{amb}^4 - T^4)$$

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 $h_u = 12.25$ W/(m^2·K) and $h_d = 6.25$ 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.

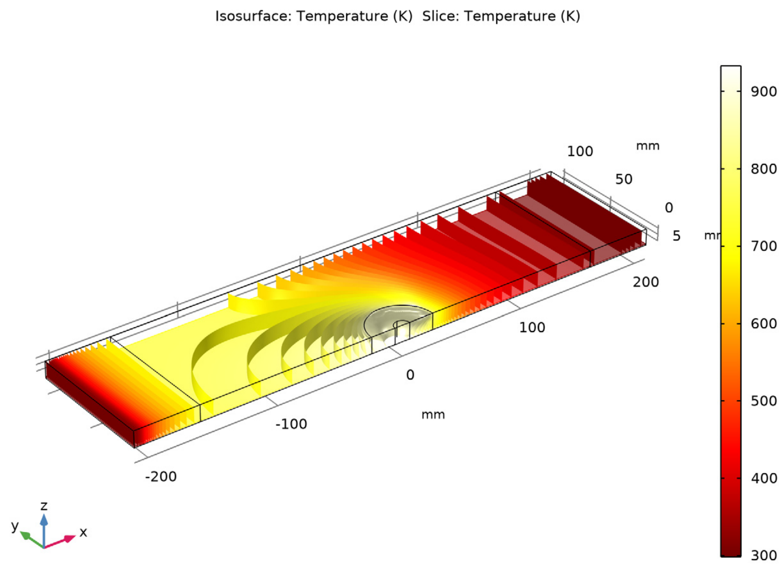


Figure 4: Temperature field in the aluminum plate.

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 **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/(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 m²	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.

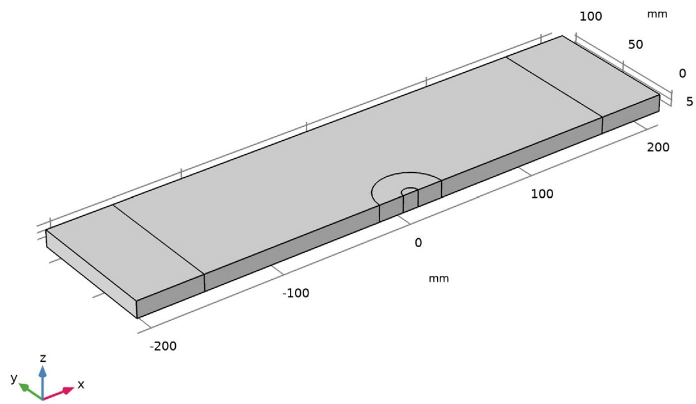
4 Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.

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 **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 In 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

Ambient Properties 1 (ampr1)

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

Set the ambient temperature to be used in 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 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, click **Solid 1**.

Translational Motion 1

1 In 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 $\mathbf{u}_{trans}$ 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 Click the **Convective heat flux** button.
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 Click the **Convective heat flux** button.
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 **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.

5 In the **Home** 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 **Boundary** 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 **Boundary** 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**.

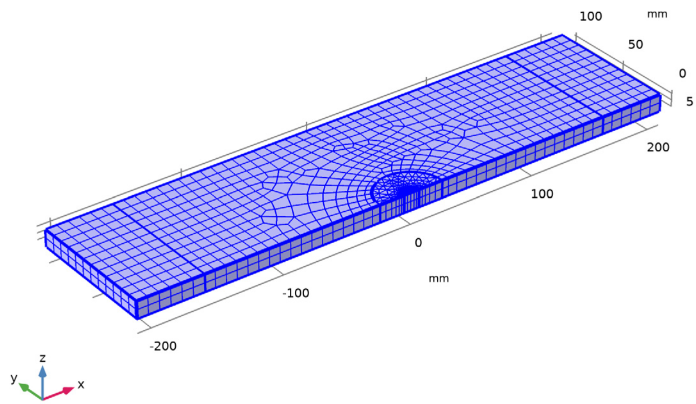
Distribution 1

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 **Home** 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)

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 **ThermalLight**.

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