COMSOL MULTIPHYSICS®

Friction Stir Welding

SOLVED WITH COMSOL MULTIPHYSICS 3.5a

COMSOL

Friction Stir Welding

Introduction

Manufacturers use a modern welding method called friction stir welding to join aluminum plates. This model 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 melts the aluminum through the generation of friction heat. The tool's rotation stirs the melted aluminum such that the two plates are joined. Figure 1 shows the rotating tool and the aluminum plates being are joined.
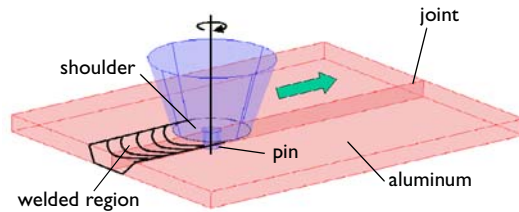
Figure 1: Two aluminum plates being joined by friction stir welding.

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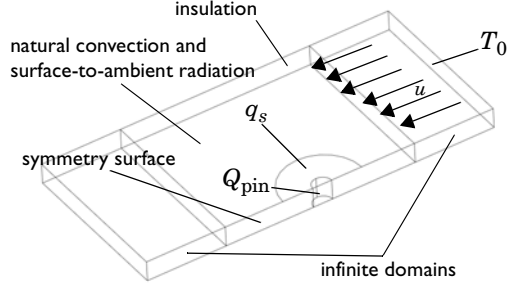

Figure 2: Model geometry for friction stir welding.

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

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

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 $u = 1.59 \cdot 10^{-3}$ 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):

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

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 model, you approximate this function with an interpolation function determined from experimental data given in Ref. 1 (see Figure 4).

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:

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

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 will be 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

$$q_{\text{up}} = h_{\text{up}}(T_0 - T) + \varepsilon\sigma(T_{\text{amb}}^4 - T^4)$$

$$q_{\text{down}} = h_{\text{down}}(T_0 - T) + \varepsilon\sigma(T_{\text{amb}}^4 - T^4)$$

where h_{up} and h_{down} 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 model, use the values $h_{\text{up}} = 12.25$ $\mathrm{W/(m^2 \cdot K)}$ and $h_{\text{down}} = 6.25$ $\mathrm{W/(m^2 \cdot K)}$

Results and Discussion

Figure 3 shows the resulting temperature field. Consider this result as what you would see through a window fixed to the moving welding tool.
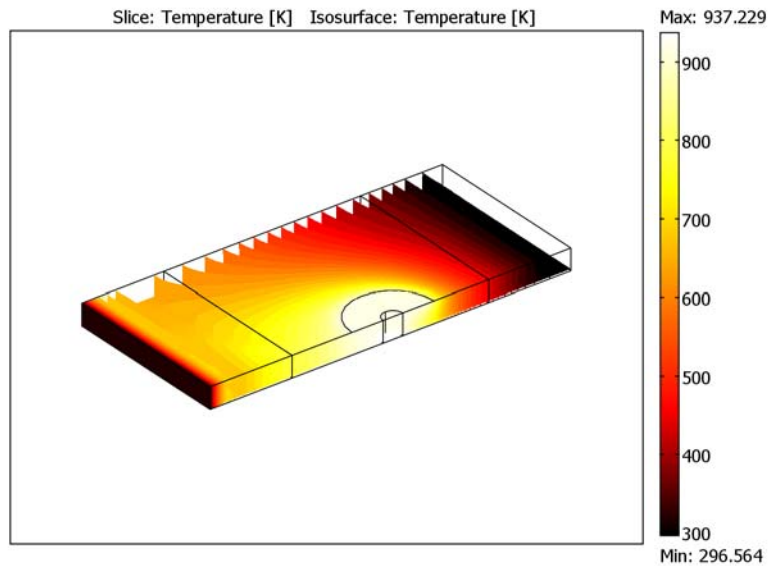

Figure 3: Temperature field in the aluminum plate.

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, *International Journal 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*, Wiley, 1993.

Model Library path: `Heat_Transfer_Module/Process_and_Manufacturing/` `friction_welding`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Open the **Model Navigator** and click the **New** tab. In the **Space dimension** list select **3D**.

2 From the list of application modes select
   Heat Transfer Module>General Heat Transfer>Steady-state analysis.

3 Click **OK**.

OPTIONS AND SETTINGS

Constants

From the **Options** menu, select **Constants**. Define the following names, expressions, and (optionally) descriptions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
T0	300[K]	Reference temperature
T_melt	933[K]	Workpiece melting temperature
rho_pin	7800[kg/m^3]	Pin density
k_pin	42[W/(m*K)]	Thermal conductivity
Cp_pin	500[J/(kg*K)]	Specific heat capacity
h_upside	12.25[W/(m^2*K)]	Heat transfer coefficient, upside
h_downside	6.25[W/(m^2*K)]	Heat transfer coefficient, downside
epsilon	0.3	Surface emissivity
u_weld	1.59[mm/s]	Welding speed
mu	0.4	Friction coefficient
n	637[1/min]	Rotation speed (RPM)
omega	2*pi[rad]*n	Angular velocity (rad/s)
F_n	25[kN]	Normal force
r_pin	6[mm]	Pin radius
r_shoulder	25[mm]	Shoulder radius
A_s	pi*(r_shoulder^2 -r_pin^2)	Shoulder surface area

Next, define an interpolation function for the aluminum yield stress, $\overline{\overline{Y}}$, as a function of the temperature, T, based on experimental material data listed in Ref. 1.

I From the **Options** menu open the **Functions** dialog box.

2 Click **New**.

3 In the **New Function** dialog box, type Ybar in the **Function name** edit field.

4 Select the **Interpolation** option button, then click **OK**.

5 Back in the **Functions** dialog box, leave the default settings for **Interpolation method** and **Extrapolation method**. In the **x** and **f(x)** columns enter the following data:

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

The **x** values are temperatures (in kelvin) and the **f** values corresponding yield stresses (in MPa) for 6061-T6 aluminum.

6 When finished, click **Plot** to view the resulting interpolation function (Figure 4), then click **OK** to close the **Functions** dialog box.
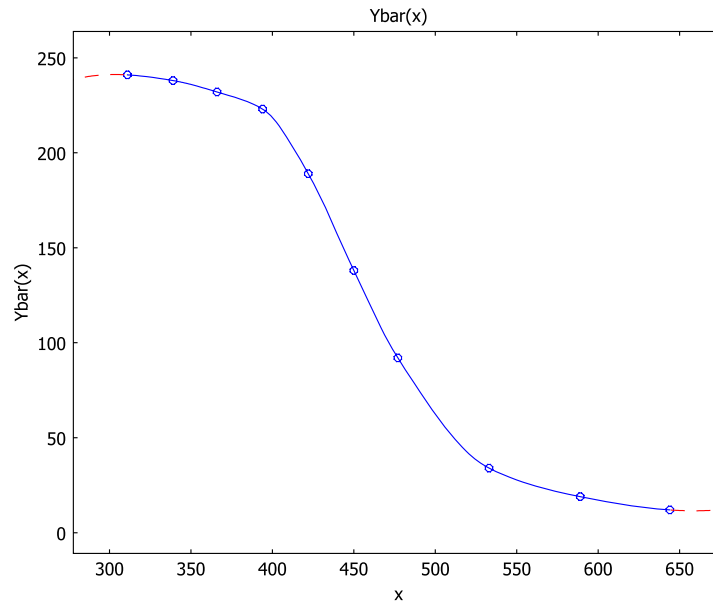

Figure 4: Yield stress (MPa) vs. temperature (K) for 6061-T6 aluminum.

GEOMETRY MODELING

1 Create the aluminum plate. From the **Draw** menu, select **Block**.

2 In the dialog box that appears, go to the **Length** area and enter the settings in the following table; when done, click **OK**.

OBJECT DIMENSIONS	EXPRESSION
X	120e-3
Y	102e-3
Z	12.7e-3

3 Repeat Steps 1 and 2 to draw another block using the settings in the following table:

OBJECT DIMENSIONS	EXPRESSION
X	50e-3
Y	102e-3
Z	12.7e-3

4 In the **Axis base point** area, type -50e-3 in the **x** edit field. Click **OK**.

5 To create a third block of the same dimensions as the one you just created, press Ctrl+C and then press Ctrl+V. In the **Paste** dialog box that appears, type 170e-3 in the **x** edit field, then click **OK**.

6 Click the **Zoom Extents** button on the Main toolbar.

7 Go to the **Draw** menu and select **Work-Plane Settings**.

8 On the **Quick** page, click the **x-y** option button, then in the **z** edit field enter 12.7e-3. Click **OK**.

9 Click the **Zoom Extents** button on the Main toolbar.

10 Press the Shift key and click the **Ellipse/Circle (Centered)** button on the Draw toolbar.

11 In the dialog box that appears, enter the following circle properties; when done, click **OK**:

OBJECT DIMENSIONS	EXPRESSION
Radius	25e-3
Base	Center
x-position	61.5e-3
y-position	0

12 In the same manner, create a second circle with the following properties:

OBJECT DIMENSIONS	EXPRESSION
Radius	6e-3
Base	Center
x-position	61.5e-3
y-position	0

13 Press the Shift key and click the **Rectangle/Square** button on the Draw toolbar.

14 In the dialog box that appears, enter the following rectangle properties; when done, click **OK**:

OBJECT DIMENSIONS	EXPRESSION
Width	50e-3
Height	25e-3
Base	Corner
x-position	36.5e-3
y-position	-25e-3

15 Create another rectangle with the following properties:

OBJECT DIMENSIONS	EXPRESSION
Width	12e-3
Height	6e-3
Base	Corner
x-position	55.5e-3
y-position	-6e-3

16 Select the circle C1 and the rectangle R1, then click the **Difference** button on the Draw toolbar.

17 Similarly, select the circle C2 and the rectangle R2, then click the **Difference** button.

18 Select the object CO1, then from the **Draw** menu select **Embed**.

19 In the dialog that appears, click **OK**.

20 Return to the **Geom2** geometry.

21 Select the object CO2, then from the **Draw** menu select **Extrude**.

22 In the dialog that appears, find the **Distance** edit field and type -12.7e-3.

23 Click **OK**.

24 Click the **Zoom Extents** button on the Main toolbar.

PHYSICS SETTINGS

Boundary Expressions

1 From the **Options** menu, select **Expressions>Boundary Expressions**.

2 Select Boundary 11, then enter the following expressions.

NAME	EXPRESSION
R	sqrt((x-0.0615)^2+y^2)
q_shoulder	(mu*F_n/A_s)*(R*omega)*flc1hs((T_melt-T)[1/K],5)

3 Select Boundaries 12 and 16, then enter the following expressions (on the third line of the table); when finished, click **OK**.

NAME	EXPRESSION
q_pin	mu/sqrt(3*(1+mu^2))*(r_pin*omega)*Ybar(T[1/K])[MPa]* flc1hs((T_melt-T)[1/K],5)

Note: The boundary expressions defined above include a smoothed step function, flc1hs, which models that the generation of friction heat is zero above the melting temperature of aluminum. Using this function is computationally more stable than multiplying the expressions by the logical expression (T<T_melt).

Subdomain Settings

1 Choose **Physics>Subdomain Settings**.

2 Select Subdomains 1, 2 and 4, then click the **Load** button.

3 From the **Materials** list, select **Basic Material Properties>Aluminum**, then click **OK**.

4 Select Subdomain 3 and enter the following settings:

PROPERTY	VALUE
k (isotropic)	k_pin
ρ	rho_pin
C_p	Cp_pin

5 Click the **Convection** tab and select Subdomains 1, 2 and 4.

6 Select the **Enable convective heat transfer** check box.

7 In the **Matter state** list, select **Solid**.

8 In the leftmost **u** edit field for the *x* velocity, type `-u_weld`.

9 Select Subdomains 1 and 4, and click the **Infinite Elements** tab.

10 From the **Type of infinite element** list, select **Cartesian**. Then select the **Stretched in x direction** check box.

11 Enter the following properties in the x_0, y_0, and z_0 edit fields:

PROPERTY	VALUE
x_0	`61.5e-3`
y_0	`3e-3`
z_0	`6e-3`

12 Select all subdomains and click the **Init** tab. Type `T0` in the **T(t_0)** edit field for the initial value.

13 Click **OK**.

Boundary Settings

1 From the **Physics** menu, select **Boundary Settings**.

2 Select the **Interior boundaries** check box.

3 Enter settings as in the following table; when finished, click **OK**.

SETTINGS	BOUNDARY 1	BOUNDARIES 3, 8, 20	BOUNDARY 4, 9, 21	BOUNDARY 11	BOUNDARIES 12, 16	BOUNDARY 23
Type	Insulation/ Symmetry	Heat flux	Heat flux	Heat flux	Heat source/ sink	Temperature
q_0				`q_shoulder`	`q_pin`	
h		`h_downside`	`h_upside`	0		
T_{inf}		`T0`	`T0`	`273.15`		
T_0						`T0`
Radiation type		Surface-to-ambient	Surface-to-ambient	None		
ε		`epsilon`	`epsilon`			
T_{amb}		`T0`	`T0`			

For the boundaries not mentioned in the table, the default setting (Insulation/ Symmetry) applies.

MESH GENERATION

From the **Mesh** menu, select **Initialize Mesh**.

COMPUTING THE SOLUTION

1 From the **Solve** menu, open the **Solver Parameters** dialog box.

2 Click the **Stationary** tab.

3 Type 60 in the **Maximum number of iterations** edit field.

4 Click **OK**.

5 From the **Solve** menu, choose **Solve Problem**.

POSTPROCESSING AND VISUALIZATION

The default plot shows a slice plot of the temperature field. To create Figure 3, which shows a slice plot and some temperature isosurfaces, follow these steps:

1 From the **Postprocessing** menu, open the **Plot Parameters** dialog box.

2 Go to the **Slice** page and find the **Slice positioning** area.

3 In the **Number of levels** edit field for **x levels**, change the value to 0.

4 From the **Color table** list in the **Slice color** area, select **Thermal**.

5 In the **Vector with coordinates** edit field for **z levels**, type `1e-3`.

6 Click **Apply**.

7 On the **Isosurface** page, select the **Isosurface plot** check box.

8 In the **Vector with isolevels** edit field, type `range(300,20,980)`.

9 From the **Color table** list in the **Isosurface color** area, select **Thermal**.

10 Clear the **Color legend** check box.

Click **OK**.