

# Fluid-Structure Interaction in Aluminum Extrusion

## Introduction

Out of all metals, the most frequently extruded is aluminum. Aluminum extrusion entails using a hydraulic ram to squeeze an aluminum bar through a die. This process will form the metal into a particular shape. Extruded aluminum is used in many manufacturing applications, such as building components for example. In massive forming processes like rolling or extrusion, metal alloys are deformed in a hot solid state with material flowing under ideally plastic conditions. Such processes can be simulated effectively using computational fluid dynamics, where the material is considered as a fluid with a very high viscosity that depends on velocity and temperature. Internal friction of the moving material acts as a heat source, so that the heat transfer equations are fully coupled with those ruling the fluid dynamics part. This approach is especially advantageous when large deformations are involved.

This model is adapted from a benchmark study in Ref. 1. The original benchmark solves a thermal-structural coupling, because it is common practice in the simulation of such processes to use specific finite element codes that have the capability to couple the structural equations with heat transfer. The alternative scheme discussed here couples non-Newtonian flow with heat transfer equations. In addition, because it is useful to know the stress in the die due to fluid pressure and thermal loads, the model adds a structural mechanics analysis.

The die design is courtesy of Compes S.p.A., while the die geometry, boundary conditions, and experimental data are taken from Ref. 1.

**Note:** This application requires the Heat Transfer Module and the Structural Mechanics Module. In addition, it uses the Material Library.

# Model Definition

The model considers steady-state conditions, assuming a billet of infinite length flowing through the die. In the actual process, the billet is pushed by the ram through the die and its volume is continuously reducing.

Figure 1 shows the original complete geometry with four different profiles. To have a model with reasonable dimensions, consider only a quarter of the original geometry. The simplification involved in neglecting the differences between the four profiles does not affect the numerical scheme proposed. Figure 2 shows the resulting model geometry.

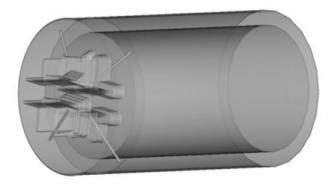


Figure 1: Original benchmark geometry.

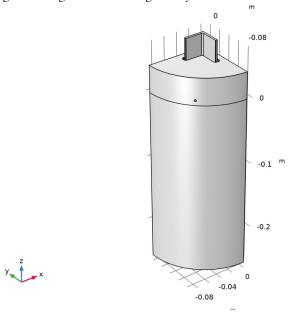


Figure 2: Quarter of the original geometry considered in the model.

#### MATERIAL PROPERTIES

The documentation for the benchmark model (Ref. 1) serves as the data source for properties of the two main materials: AISI steel for the die and the container (the ram is not considered here) and aluminum for the billet.

## Structural Analysis

Because only the steel part is active in the structural analysis, consider a simple linear elastic behavior where the elastic properties are those of the material H11 mod (AISI 610) that can be found in the COMSOL Multiphysics Material Library.

# Heat Transfer Analysis

The benchmark model uses the following properties for aluminum and steel:

ALUMINUM	1 VALUE DESCRIPTION		
$k_{ m al}$	210 N/(s·K)	Thermal conductivity	
$ ho_{al}$	2700 kg/m <sup>3</sup>	Density	
$C_{ m pal}$	2.94 N/(mm $^2$ ·K)/ $\rho_{al}$	Heat capacity at constant pressure	
-			
STEEL	VALUE	DESCRIPTION	
STEEL $k_{ m fe}$	VALUE 24.33 N/(s·K)	DESCRIPTION  Thermal conductivity	

#### Non-Newtonian Flow

The properties of the aluminum were experimentally determined and then checked using literature data for the same alloy and surface state. However the benchmark proposes an experimental constitutive law, suited for the structural mechanics codes usually used to simulate such processes, in the form of the flow stress data. For this model this requires a recalculation of the constitutive law to derive a general expression for the viscosity. The equivalent von Mises stress,  $\sigma_{eqv}$ , can be defined in terms of the total contraction of the deviatoric stress tensor as

$$\sigma_{\rm eqv} = \sqrt{\frac{3}{2}\tau : \tau}$$

or, using  $\tau = 2\eta \dot{\epsilon}$  where  $\dot{\epsilon}$  is the strain rate and  $\eta$  is the viscosity, as

$$\sigma_{\rm eqv} = \sqrt{6\eta^2 \dot{\epsilon} : \dot{\epsilon}}$$
 (1)

Introducing the equivalent strain rate

$$\dot{\phi}_{eqv} \equiv \sqrt{\frac{2}{3}\dot{\epsilon}:\dot{\epsilon}}$$

Equation 1 can be expressed as

$$\sigma_{\rm eqv} = 3\eta \dot{\phi}_{\rm eqv}$$

The strain rate tensor is defined as (Ref. 2)

$$\dot{\varepsilon} = \frac{\nabla \mathbf{u} + (\nabla \mathbf{u})^T}{2} = \frac{1}{2}\dot{\gamma}$$

The shear rate  $\dot{\gamma}$  is defined as

$$\dot{\gamma} = |\dot{\gamma}| = \sqrt{\frac{1}{2}\dot{\gamma}:\dot{\gamma}}$$

so that

$$\phi_{\rm eqv} = \frac{1}{\sqrt{3}} \dot{\gamma}$$

The flow rule

$$\sigma_{\rm eqv} = \kappa_{\rm f}$$

states that plastic yielding occurs if the equivalent stress,  $\sigma_{eqv}$ , reaches the flow stress,  $\kappa_f$ . The viscosity is defined as (see Ref. 2 for further details)

$$\eta = \frac{\kappa_{\rm f}}{3\dot{\phi}_{\rm eqv}}$$

The organizers of the benchmark propose specific flow-stress data expressed in terms of a generalized Zener-Hollomon function

$$\eta = \frac{a sinh\left(\left(\frac{Z}{A}\right)^{\frac{1}{n}}\right)}{\sqrt{3}\alpha\dot{\gamma}}$$

where  $A = 2.39 \cdot 10^8 \text{ s}^{-1}$ , n = 2.976,  $\alpha = 0.052 \text{ MPa}^{-1}$ , and

$$Z = \frac{1}{\sqrt{3}} \dot{\gamma} e^{\left(\frac{Q}{RT}\right)}$$

with Q = 153 kJ/mol and R = 8.314 J/(K·mol).

## SOURCES, INITIAL CONDITIONS, AND BOUNDARY CONDITIONS

# Structural Analysis

Because the model geometry is a quarter of the actual geometry, use symmetric boundary conditions for the two orthogonal planes. On the external surfaces of the die, apply roller boundary conditions because in reality other dies, not considered here, are present to increase the system's stiffness.

The main loads are the thermal loads from the heat transfer analysis and the total stress from the fluid dynamics analysis.

# Heat Transfer Analysis

For the billet, use a volumetric heat source related to the viscous heating effect.

The external temperature of the ram and the die is held constant at 450°C (723 K). The ambient temperature is 25°C (298 K). For the heat exchange between aluminum and steel, use the heat transfer coefficient of 11 N/(s·mm·K). Also consider convective heat exchange with air outside the profiles with a fixed convective heat transfer coefficient of 15 W/( $m^2 \cdot K$ ).

Apply initial temperatures as given in the following table:

PART	VALUE
Ram	380°C (653 K)
Container	450°C (723 K)
Billet	460°C (733 K)
Die	404°C (677 K)

## Non-Newtonian Flow

At the inlet, the ram moves with a constant velocity of 0.5 mm/s. Impose this boundary condition by simply applying a constant inlet velocity. At the outlet, a normal stress condition with zero external pressure applies. On the surfaces placed on the two symmetry planes, use symmetric conditions. Finally, apply slip boundary conditions on the boundaries placed outside the profile.

The general response of the proposed numerical scheme, especially in the zone of the profile, is in good accordance with the experience of the designers. A comparison between the available experimental data and the numerical results of the simulation shows good agreement.

On the basis of the results from the simulation, the engineer can improve the preliminary die design by adjusting relevant physical parameters and operating conditions. For this purpose, the volume plot in Figure 3 showing the temperature field inside the profile gives important information. Furthermore, the combined streamline and slice plot in Figure 4 reveals any imbalances in the velocity field that could result in a crooked profile. A proper design should also ensure that different parts of the profile travel at the same speed. Figure 5 shows the von Mises equivalent stress in the steel part considering the thermal load and the pressure load due to the presence of the fluid.

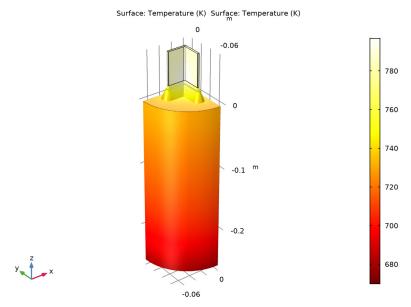


Figure 3: Temperature distribution in the billet.

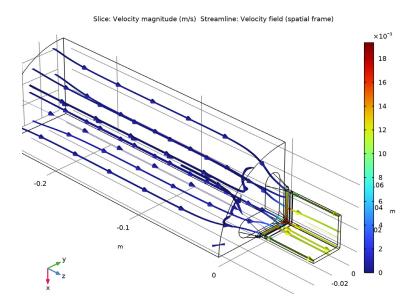


Figure 4: Velocity field and streamlines at the profile section.

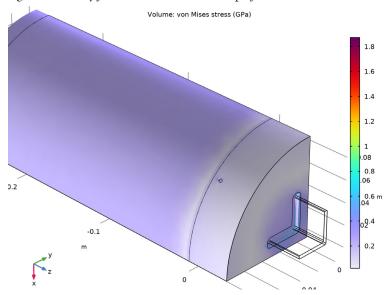


Figure 5: Equivalent von Mises stress distribution in the container.

- 1. M. Schikorra, L. Donati, L. Tomesani, and A.E. Tekkaya, "The Extrusion Benchmark 2007," *Proceedings of the Extrusion Workshop 2007 and 2nd Extrusion Benchmark Conference*, Bologna, Italy, http://diemtech.ing.unibo.it/extrusion07.
- 2. E.D. Schmitter, "Modelling massive forming processes with thermally coupled fluid dynamics," *Proceedings of the COMSOL Multiphysics User's Conference 2005 Frankfurt*, Frankfurt, Germany.

**Application Library path:** Heat\_Transfer\_Module/Thermal\_Processing/aluminum\_extrusion\_fsi

# Modeling Instructions

From the File menu, choose New.

#### NEW

In the New window, click Model Wizard.

#### MODEL WIZARD

- I In the Model Wizard window, click **3D**.
- 2 In the Select Physics tree, select Heat Transfer>Conjugate Heat Transfer>Laminar Flow.
- 3 Click Add.
- 4 In the Select Physics tree, select Structural Mechanics>Solid Mechanics (solid).
- 5 Click Add.
- 6 Click Study.
- 7 In the Select Study tree, select General Studies>Stationary.
- 8 Click **Done**.

#### **GEOMETRY I**

Import I (impl)

- I In the Home toolbar, click Import.
- 2 In the Settings window for Import, locate the Import section.
- 3 Click Browse.

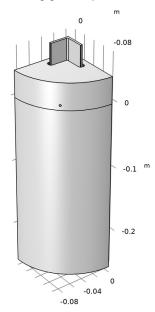
- 4 Browse to the model's Application Libraries folder and double-click the file aluminum\_extrusion\_fsi.mphbin.
- 5 Click Import.
- 6 Click the Zoom Extents button in the Graphics toolbar.

The imported geometry contains narrow face regions, which are not necessary for this model and will increase the number of elements significantly. Add a Remove Details feature to remove these details.

Remove Details I (rmd1)

- I In the Geometry toolbar, click Remove Details.
- 2 Click III Build All.

You should now see the following geometry.





#### **GLOBAL DEFINITIONS**

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, locate the Parameters section.

# **3** In the table, enter the following settings:

Name	Expression	Value	Description
D_alfe	1 [ mm ]	0.001 m	Thickness of the high conductive layer
Heat_alfe	11[N/(s*mm*K)]	11000 W/(m <sup>2</sup> ·K)	Aluminum-steel heat exchange coefficient
T_billet	460[degC]	733.15 K	Billet temperature
T_container	450[degC]	723.15 K	Container temperature
T_ram	380[degC]	653.15 K	Ram temperature
T_pd1	404[degC]	677.15 K	Initial temperature around thermocouple at point PD1
V_ram	0.5[mm/s]	5E-4 m/s	Ram velocity
P_init	0[bar]	0 Pa	External reference pressure
T_air	25[degC]	298.15 K	Ambient temperature
Q_eta	153000[J/mol]	1.53E5 J/mol	Parameter Q for the generalized Zener-Hollomon function
n_eta	2.976	2.976	Parameter n for the generalized Zener- Hollomon function
A_eta	2.39e8[1/s]	2.39E8 1/s	Parameter A for the generalized Zener- Hollomon function
alpha_eta	0.0521[1/MPa]	5.21E-8 I/Pa	Parameter alpha for the generalized Zener- Hollomon function
H_conv	15	15	Convective heat exchange coefficient with air
F	sqrt(1/3)	0.57735	Factor for the conversion of the shear rate to COMSOL's definition

# DEFINITIONS

Variables I

- I In the Home toolbar, click a= Variables and choose Local Variables.
- 2 In the Settings window for Variables, locate the Variables section.

**3** In the table, enter the following settings:

Name	Expression	Unit	Description
Z_eta	F*spf.sr*exp(Q_eta/(R_const*T))	I/s	Zener-Hollomon parameter
mu_al	<pre>asinh((Z_eta/A_eta)^(1/ n_eta))/(3*alpha_eta*F* spf.sr+sqrt(eps))</pre>	Pa∙s	Viscosity of aluminum

Create the selections to simplify the model specification.

#### Outside

- I In the **Definitions** toolbar, click **\( \frac{1}{2} \) Explicit**.
- 2 In the Settings window for Explicit, type Outside in the Label text field.
- 3 Locate the Input Entities section. From the Geometric entity level list, choose Boundary.
- **4** Select Boundaries 35–38, 42, 43, 50, 51, 53, 55, 70, 71, 79–82, 87, 88, 93, 95, 102, 103, 106, and 108 only.

For more convenience in selecting these boundaries, you can click the Paste Selection button and paste the above numbers.

#### Interior

- I In the **Definitions** toolbar, click **\( \) Explicit**.
- 2 In the Settings window for Explicit, type Interior in the Label text field.
- 3 Locate the Input Entities section. From the Geometric entity level list, choose Boundary.
- **4** Select Boundaries 8, 11, 14, 15, 19, 20, 24, 29–34, 41, 44, 45, 49, 52, 58–60, 64, 69, 72, 73, 76–78, 86, 89–92, 101, 104, 105, and 109 only.

Before creating the materials for the model, specify the fluid and solid domains. Using this information, the software can detect which material properties are needed.

#### HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

#### Fluid 1

- I In the Model Builder window, under Component I (compl)> Heat Transfer in Solids and Fluids (ht) click Fluid 1.
- 2 Select Domains 3 and 4 only.
- 3 In the Settings window for Fluid, locate the Domain Selection section.
- 4 Click **Greate Selection**.
- 5 In the Create Selection dialog box, type Billet in the Selection name text field.

#### 6 Click OK.

## LAMINAR FLOW (SPF)

- I In the Model Builder window, under Component I (compl) click Laminar Flow (spf).
- 2 In the Settings window for Laminar Flow, locate the Physical Model section.
- 3 From the Compressibility list, choose Incompressible flow.
- 4 Locate the **Domain Selection** section. From the **Selection** list, choose **Billet**.

## SOLID MECHANICS (SOLID)

- I In the Model Builder window, under Component I (compl) click Solid Mechanics (solid).
- 2 In the Settings window for Solid Mechanics, locate the Domain Selection section.
- 3 Click Clear Selection.
- **4** Select Domains 1 and 2 only.

Now, define the material for each domain.

#### ADD MATERIAL

- I 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>Structural steel.
- 4 Click Add to Component in the window toolbar.
- 5 In the Home toolbar, click 4 Add Material to close the Add Material window.

#### MATERIALS

Structural steel (mat I)

- I Select Domains 1 and 2 only.
- 2 In the Settings window for Material, locate the Material Contents section.

**3** In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Heat capacity at constant pressure	Ср	4.63[N/ (mm^2* K)]/ rho(T[1/ K])[kg/ m^3]	J/(kg·K)	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	24.33[N/ (s*K)]	W/(m·K)	Basic
Poisson's ratio	nu	0.3	I	Young's modulus and Poisson's ratio

The heat capacity is not used since it does not enter in the stationary heat transfer equation for solids without translational movement. The heat capacity is only provided for completeness in case you want to extend the model to perform transient simulations.

#### Billet

- I In the Materials toolbar, click **Blank Material**.
- 2 In the Settings window for Material, type Billet in the Label text field.
- 3 Locate the Geometric Entity Selection section. From the Selection list, choose Billet.
- **4** Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Heat capacity at constant pressure	Ср	2.94[N/(mm^2*K)]/rho	J/(kg·K)	Basic
Density	rho	2700	kg/m³	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	210	W/(m·K)	Basic
Dynamic viscosity	mu	mu_al	Pa·s	Basic

## Structural steel 1 (mat3)

- I In the Model Builder window, under Component I (compl)>Materials right-click Structural steel (mat I) and choose Duplicate.
- 2 In the Settings window for Material, locate the Geometric Entity Selection section.

- 3 From the Geometric entity level list, choose Boundary.
- 4 From the Selection list, choose Interior.

Now, define the volume reference temperature in the solid domains. This temperature is used by materials to define the density in solids for both the **Heat Transfer in Solids and Fluids** and **Solid Mechanics** interfaces, and by the **Thermal Expansion** feature as the zero strain reference.

#### DEFINITIONS

#### Model Inbut 1

- I In the Physics toolbar, click **Shared Properties** and choose **Model Input**.
- 2 Select Domain 1 only.
- 3 In the Settings window for Model Input, locate the Definition section.
- 4 Click Select Quantity.
- 5 In the Physical Quantity dialog box, type temperature in the text field.
- 6 Click **Filter**.
- 7 In the tree, select General>Volume reference temperature (K).
- 8 Click OK.
- 9 In the Settings window for Model Input, locate the Definition section.
- 10 In the text field, type T container.

This value overrides the value defined under **Default Model Inputs** for the selected domain.

#### SHARED PROPERTIES

## Model Input 2

- I In the Physics toolbar, click **Shared Properties** and choose **Model Input**.
- 2 Select Domain 2 only.
- 3 In the Settings window for Model Input, locate the Definition section.
- 4 Click Select Quantity.
- 5 In the Physical Quantity dialog box, select General>Volume reference temperature (K) in the tree.
- 6 Click OK.
- 7 In the Settings window for Model Input, locate the Definition section.
- 8 In the text field, type T pd1.

With the materials defined, you can set up the remaining physics of the model.

# LAMINAR FLOW (SPF)

In the current model, the high viscosity of the fluid flow implies that the model is diffusion dominated. Pseudo time stepping is thus not adapted to this case as it is based on the scale of the convective flux.

- I Click the Show More Options button in the Model Builder toolbar.
- 2 In the Show More Options dialog box, in the tree, select the check box for the node Physics>Advanced Physics Options.
- 3 Click OK.
- 4 In the Model Builder window, under Component I (compl) click Laminar Flow (spf).
- 5 In the Settings window for Laminar Flow, click to expand the Advanced Settings section.
- **6** Find the **Pseudo time stepping** subsection. From the Use pseudo time stepping for stationary equation form list, choose Off.

Initial Values 1

- I In the Model Builder window, under Component I (compl)>Laminar Flow (spf) click Initial Values 1.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the *p* text field, type P init.

Symmetry I

- I In the Physics toolbar, click **Boundaries** and choose Symmetry.
- 2 Select Boundaries 9 and 114 only.

Inlet I

- I In the Physics toolbar, click **Boundaries** and choose Inlet.
- 2 Select Boundary 10 only.
- 3 In the Settings window for Inlet, locate the Velocity section.
- 4 Click the **Velocity field** button.
- **5** Specify the  $\mathbf{u}_0$  vector as

0	x
0	у
V_ram	z

#### Wall 2

- I In the Physics toolbar, click Boundaries and choose Wall.
- 2 In the Settings window for Wall, locate the Boundary Selection section.
- **3** From the **Selection** list, choose **Outside**.
- 4 Locate the Boundary Condition section. From the Wall condition list, choose Slip.

#### Outlet 1

- I In the Physics toolbar, click **Boundaries** and choose **Outlet**.
- 2 Select Boundary 40 only.
- 3 In the Settings window for Outlet, locate the Pressure Conditions section.
- **4** In the  $p_0$  text field, type P\_init.

# HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

#### Initial Values 1

- I In the Model Builder window, under Component I (compl)>
  Heat Transfer in Solids and Fluids (ht) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the *T* text field, type T\_container.

#### Temperature I

- I In the Physics toolbar, click **Boundaries** and choose **Temperature**.
- **2** Select Boundaries 2, 5, and 7 only.
- 3 In the Settings window for Temperature, locate the Temperature section.
- **4** In the  $T_0$  text field, type T\_container.

## Heat Flux I

- I In the Physics toolbar, click **Boundaries** and choose **Heat Flux**.
- **2** Select Boundary 10 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 Heat alfe.
- **6** In the  $T_{\text{ext}}$  text field, type T\_ram.

#### Heat Flux 2

- I In the Physics toolbar, click **Boundaries** and choose **Heat Flux**.
- 2 In the Settings window for Heat Flux, locate the Boundary Selection section.

- **3** From the **Selection** list, choose **Outside**.
- 4 Locate the Heat Flux section. From the Flux type list, choose Convective heat flux.
- **5** In the *h* text field, type H conv.
- **6** In the  $T_{\rm ext}$  text field, type T\_air.

#### Outflow I

- I In the Physics toolbar, click **Boundaries** and choose **Outflow**.
- **2** Select Boundary 40 only.

#### Thin Layer I

- I In the Physics toolbar, click **Boundaries** and choose **Thin Layer**.
- 2 In the Settings window for Thin Layer, locate the Boundary Selection section.
- 3 From the Selection list, choose Interior.
- 4 Locate the Layer Model section. From the Specify list, choose Thermal resistance.
- **5** Locate the **Heat Conduction** section. In the  $R_s$  text field, type 1/Heat\_alfe.

#### MATERIALS

Structural steel 1 (mat3)

- I In the Model Builder window, under Component I (compl)>Materials click Structural steel I (mat3).
- 2 In the Settings window for Material, locate the Material Contents section.
- **3** In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thickness	lth	D_alfe	m	Shell

#### SOLID MECHANICS (SOLID)

For faster convergence use linear elements. You can always improve the solution using the default quadratic elements.

- I In the Model Builder window, under Component I (compl) click Solid Mechanics (solid).
- 2 In the Settings window for Solid Mechanics, click to expand the Discretization section.
- 3 From the Displacement field list, choose Linear.

## Linear Elastic Material I

Both Young's modulus and Poisson's ratio are temperature dependent. Follow these steps to verify that the temperature is defined by the Heat Transfer in Solids and Fluids interface.

- I In the Model Builder window, under Component I (compl)>Solid Mechanics (solid) click Linear Elastic Material I.
- 2 In the Settings window for Linear Elastic Material, locate the Model Input section.
- 3 Click **Go to Source** for **Temperature** for the **Temperature** (T).

#### **GLOBAL DEFINITIONS**

Default Model Inputs

Locate the **Browse Model Inputs** section. The icon column of the table under **Model input contributions** shows that it is the **Heat Transfer in Solids and Fluids** interface that defines the temperature in the solid and fluid domains and on the thin-layer boundaries.

# SOLID MECHANICS (SOLID)

Roller I

- I In the Physics toolbar, click **Boundaries** and choose Roller.
- **2** Select Boundaries 2, 5, and 7 only.

Symmetry I

- I In the Physics toolbar, click **Boundaries** and choose Symmetry.
- 2 Select Boundaries 1, 4, 112, and 113 only.

#### MULTIPHYSICS

Fluid-Structure Interaction 1 (fsi1)

- I In the Physics toolbar, click Multiphysics Couplings and choose Boundary>Fluid-Structure Interaction.
- 2 In the Settings window for Fluid-Structure Interaction, locate the Boundary Selection section.
- 3 From the Selection list, choose All boundaries.

Thermal Expansion 1 (tel)

- I In the Physics toolbar, click Multiphysics Couplings and choose Domain>
  Thermal Expansion.
- 2 Select Domains 1 and 2 only.

#### MESH I

Free Triangular 1

I In the Mesh toolbar, click A Boundary and choose Free Triangular.

2 Select Boundary 40 only.

#### Size 1

- I Right-click Free Triangular I and choose Size.
- 2 In the Settings window for Size, locate the Element Size section.
- **3** Click the **Custom** button.
- 4 Locate the Element Size Parameters section.
- 5 Select the Maximum element size check box. In the associated text field, type 0.0014.
- **6** Select the **Curvature factor** check box. In the associated text field, type **0.2**.
- 7 Click Build Selected.

# Swebt I

- I In the Mesh toolbar, click A Swept.
- 2 In the Settings window for Swept, locate the Domain Selection section.
- 3 From the Geometric entity level list, choose Domain.
- 4 Select Domain 4 only.

#### Distribution 1

- I Right-click Swept I and choose Distribution.
- 2 In the Settings window for Distribution, locate the Distribution section.
- 3 In the Number of elements text field, type 24.
- 4 Click Build All.

#### Free Tetrahedral I

In the Mesh toolbar, click A Free Tetrahedral.

#### Size 1

- I Right-click Free Tetrahedral I and choose Size.
- 2 In the Settings window for Size, locate the Element Size section.
- **3** Click the **Custom** button.
- 4 Locate the Element Size Parameters section.
- 5 Select the Maximum element size check box. In the associated text field, type 0.0085.

#### Size 2

- I In the Model Builder window, right-click Free Tetrahedral I and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Boundary.

- 4 Select Boundaries 12 and 13 only.
- **5** Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the Element Size Parameters section.
- 7 Select the Maximum element size check box. In the associated text field, type 0.002.

#### Size 3

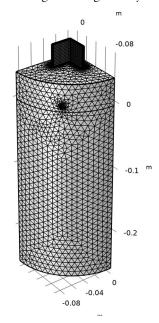
- I Right-click Free Tetrahedral I and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Domain.
- 4 Select Domain 3 only.
- **5** Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the Element Size Parameters section.
- 7 Select the Minimum element size check box. In the associated text field, type 5e-4.

# Size

- I In the Model Builder window, under Component I (compl)>Mesh I click Size.
- 2 In the Settings window for Size, locate the Element Size section.
- **3** From the **Predefined** list, choose **Finer**.

# 4 Click Build All.

You should now see the following meshed geometry.





#### STUDY I

## Step 1: Stationary

Use two stationary study steps. Solve first for the fluid dynamics and heat transfer to determine the thermal load and the pressure load and then for the structural mechanics.

- I In the Model Builder window, under Study I click Step I: Stationary.
- 2 In the Settings window for Stationary, locate the Physics and Variables Selection section.
- 3 In the table, clear the Solve for check box for Solid Mechanics (solid).
- 4 In the table, clear the Solve for check boxes for Fluid-Structure Interaction 1 (fsi1) and Thermal Expansion I (tel).

# Stationary 2

- I In the Study toolbar, click Study Steps and choose Stationary>Stationary.
- 2 In the Settings window for Stationary, locate the Physics and Variables Selection section.
- 3 In the table, clear the Solve for check boxes for Heat Transfer in Solids and Fluids (ht) and Laminar Flow (spf).

4 In the table, clear the Solve for check box for Nonisothermal Flow I (nitf1).

For the structural analysis, use a memory efficient iterative solver to make it possible to solve the problem also on computers with limited memory.

Solution I (soll)

- I In the Study toolbar, click Show Default Solver.
- 2 In the Model Builder window, expand the Solution I (soll) node.
- 3 In the Model Builder window, expand the Study I>Solver Configurations> Solution I (solI)>Stationary Solver 2 node.
- 4 Right-click Study I>Solver Configurations>Solution I (solI)>Stationary Solver 2 and choose Iterative.
- 5 In the Study toolbar, click **Compute**.

#### RESULTS

Temperature (ht)

The first default plot shows the temperature (Figure 3).

- I In the Settings window for 3D Plot Group, locate the Data section.
- 2 From the Dataset list, choose Exterior Walls.

Surface 2

- I In the Model Builder window, expand the Temperature (ht) node, then click Surface 2.
- 2 In the Settings window for Surface, click to expand the Title section.
- **3** From the **Title type** list, choose **None**.
- 4 In the Temperature (ht) toolbar, click Plot.

Modify the third default plot to see the velocity field and streamlines at the profile section (Figure 4).

Study I/Solution Store I (sol2)

In the Model Builder window, expand the Results>Datasets node, then click Study I/ Solution Store I (sol2).

Selection

- I In the Results toolbar, click 🔓 Attributes and choose Selection.
- 2 In the Settings window for Selection, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Domain.
- **4** From the **Selection** list, choose **Billet**.

#### Velocity (spf)

- I In the Model Builder window, under Results click Velocity (spf).
- 2 In the Settings window for 3D Plot Group, locate the Data section.
- 3 From the Dataset list, choose Study I/Solution Store I (sol2).

#### Slice

- I In the Model Builder window, expand the Velocity (spf) node, then 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 0.0151.
- 6 In the Velocity (spf) toolbar, click Plot.

# Velocity (sbf)

In the Model Builder window, click Velocity (spf).

#### Streamline 1

- I In the Velocity (spf) toolbar, click **Streamline**.
- 2 In the Settings window for Streamline, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>Laminar Flow> Velocity and pressure>u,v,w - Velocity field (spatial frame).
- 3 Locate the Streamline Positioning section. From the Positioning list, choose Magnitude controlled.
- 4 In the Minimum distance text field, type 0.01.
- 5 In the Maximum distance text field, type 0.1.
- 6 Locate the Coloring and Style section. Find the Line style subsection. From the Type list, choose Ribbon.
- 7 In the Width expression text field, type 0.001.
- 8 Select the Width scale factor check box.
- **9** Find the **Point style** subsection. From the **Type** list, choose **Arrow**.
- **10** Select the **Number of arrows** check box. In the associated text field, type 70.
- II Click to expand the Inherit Style section.

## Color Expression 1

I In the Velocity (spf) toolbar, click (2) Color Expression.

2 In the Settings window for Color Expression, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)> Laminar Flow>Velocity and pressure>spf.U - Velocity magnitude - m/s.

#### Streamline 1

- I In the Model Builder window, click Streamline I.
- 2 In the Settings window for Streamline, locate the Inherit Style section.
- 3 From the Plot list, choose Slice.
- 4 In the Velocity (spf) toolbar, click Plot.

To get a better view, rotate the geometry in the **Graphics** window. You can preserve a view for a plot by creating a **View** feature node as follows:

- 5 Click the **Show More Options** button in the **Model Builder** toolbar.
- 6 In the Show More Options dialog box, in the tree, select the check box for the node Results>Views.
- 7 Click OK.

#### View 3D 3

- I In the Model Builder window, under Results right-click Views and choose View 3D.
- **2** Use the **Graphics** toolbox to get a satisfying view.
- 3 In the Settings window for View 3D, locate the View section.
- 4 Select the Lock camera check box.

Next, apply the view to the velocity plot.

## Velocity (spf)

- I In the Model Builder window, under Results click Velocity (spf).
- 2 In the Settings window for 3D Plot Group, locate the Plot Settings section.
- 3 From the View list, choose View 3D 3.
- 4 In the Velocity (spf) toolbar, click Plot.

For a clearer visualization, you can duplicate this plot group and remove the geometry edges and the velocity slice, and add instead a temperature surface at the outside part.

## Velocity and Outside Temperature

- I Right-click Velocity (spf) and choose Duplicate.
- 2 In the Settings window for 3D Plot Group, type Velocity and Outside Temperature in the Label text field.
- 3 Right-click Velocity and Outside Temperature and choose Move Up.

4 Right-click Velocity and Outside Temperature and choose Move Up.

Slice

- I In the Model Builder window, expand the Velocity and Outside Temperature node.
- 2 Right-click Results>Velocity and Outside Temperature>Slice and choose Delete.
- **3** Click **Yes** to confirm.

Surface 2

- I In the Results toolbar, click More Datasets and choose Surface.
- 2 Select Boundaries 55, 95, and 108 only.

Velocity and Outside Temperature

In the Model Builder window, under Results click Velocity and Outside Temperature.

#### Surface 1

- I In the Velocity and Outside Temperature toolbar, click Surface.
- 2 In the Settings window for Surface, locate the Data section.
- 3 From the Dataset list, choose Surface 2.
- 4 Locate the Coloring and Style section. Click Change Color Table.
- 5 In the Color Table dialog box, select Thermal>HeatCameraLight in the tree.
- 6 Click OK.

Velocity and Outside Temperature

- I In the Model Builder window, click Velocity and Outside Temperature.
- 2 In the Settings window for 3D Plot Group, locate the Color Legend section.
- 3 From the Position list, choose Right double.
- 4 Locate the Plot Settings section. Clear the Plot dataset edges check box.
- 5 In the Velocity and Outside Temperature toolbar, click Plot.

Stress (solid)

The last plot shows the von Mises stress and deformation distribution in the container. To reproduce the Figure 5, apply the View 3D 3.

- I In the Model Builder window, click Stress (solid).
- 2 In the Settings window for 3D Plot Group, locate the Plot Settings section.
- 3 From the View list, choose View 3D 3.

Volume 1

I In the Model Builder window, expand the Stress (solid) node, then click Volume I.

- 2 In the Settings window for Volume, locate the Expression section.
- 3 From the Unit list, choose GPa.
- 4 In the Stress (solid) toolbar, click Plot.