Bimetallic Strip in Airflow

Introduction

This is a conceptual example, illustrating how to couple fluid-structure interaction, heat transfer, and thermal expansion.

A bimetallic strip in an air channel is heated so that it bends. After some time, an airflow with an inlet temperature that varies with time is introduced. As a result, there are changes in deformation caused by both the fluid pressure and the convective cooling. The computed structural deformations are compared for three different cases:

• No imposed airflow through the channel.

• Convective cooling with an airflow at room temperature.

• Hot airflows through the channel.

Model Definition

The bimetallic strip geometry consists of two 10 mm-by-0.5 cm strips made of copper (lower part) and iron (upper part) materials. The bimetallic strip is placed inside a 70 cm-by-10 cm air channel.
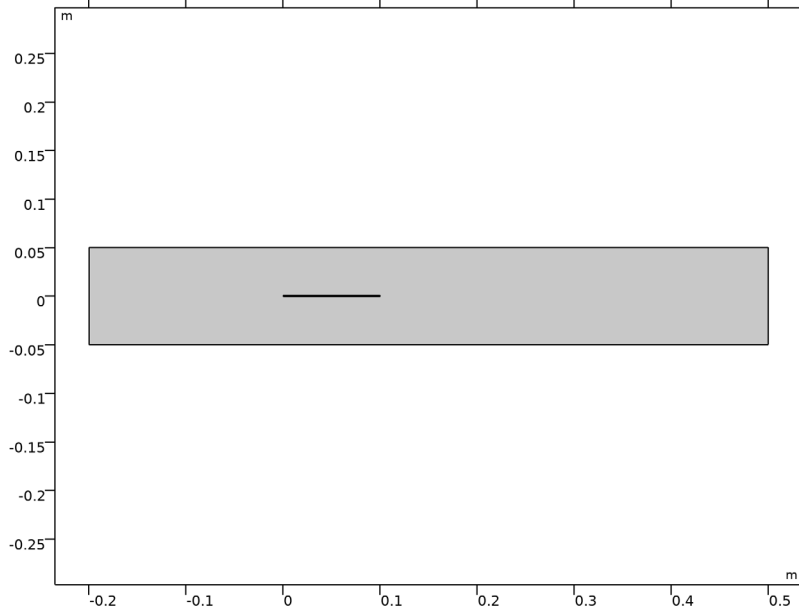

Figure 1: Model geometry.

The strip displacements are fixed at the left (upstream) extremity. A 50 W heat source with a uniform distribution is applied to the structure. When the strip is heated, the copper expands more than the iron due to the difference in thermal expansion coefficients. This causes the strip to bend upward.

At the air channel inlet (on the left), the velocity varies with time. Initially there is no flow. After 3000 seconds the inlet velocity is smoothly ramped up to 0.2 m/s.

The inlet temperature is initially set to room temperature (293.15 K). After 5000 seconds it is ramped up to 593.15 K.

Results and Discussion

Figure 2 shows the vertical displacement at the tip of the bimetallic strip as function of time.

- From 0 to 3000 seconds, the tip displacement keeps increasing as there is only a very weak cooling caused by self-convection.
- Between 3000 and 5000 seconds, there is a decay is the tip displacement as a forced convective cooling is introduced. A steady-state solution is observed, where the tip displacement is about 2 mm.
- Between 5000 and 7000 seconds, the air is heated, and the tip displacement increases again. A new steady-state is reached, at which there is a balance between the heat generated in the strip and the heat dissipated by convection.
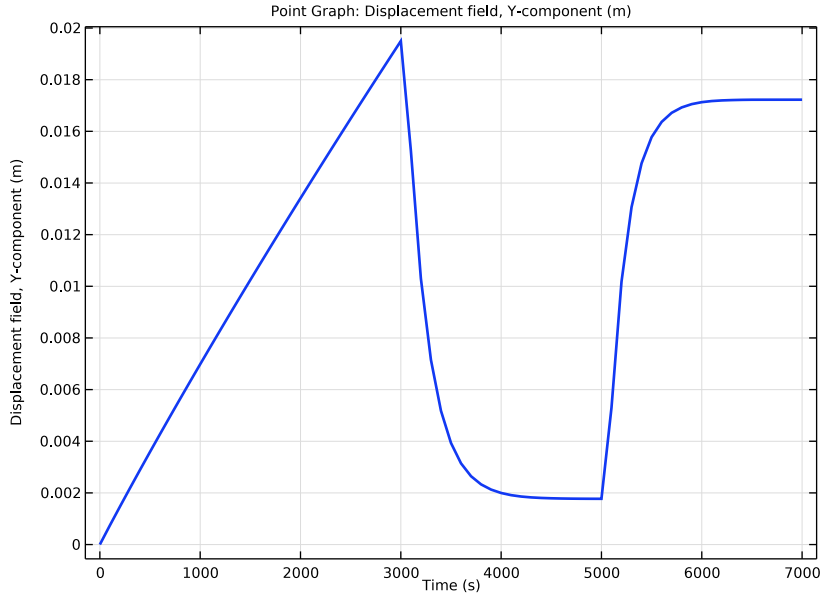
Point Graph: Displacement field, Y-component (m)

Figure 2: Vertical strip tip displacement.

Figure 3 shows the temperature distribution in the channel when there is no imposed airflow.
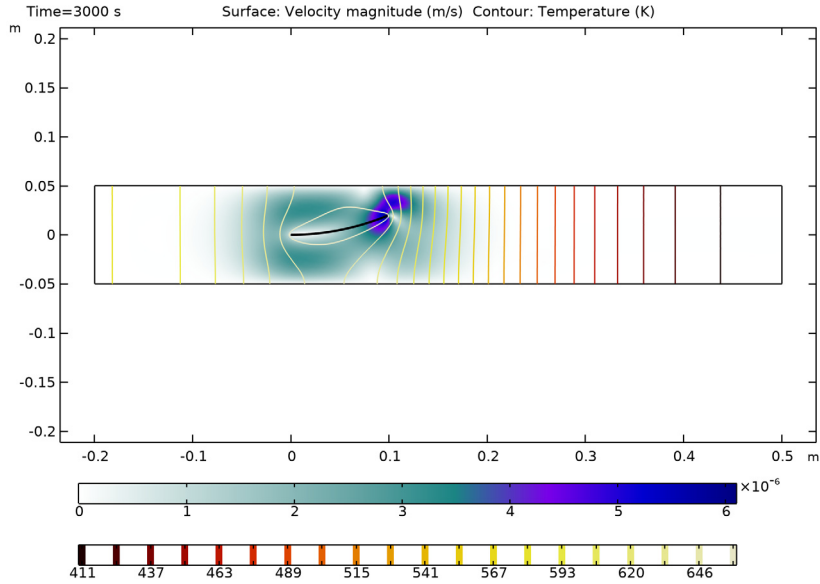
Figure 3: Velocity (surface) and temperature (contour) in the channel at 3000 s.

Figure 4 shows the temperature distribution in the channel with an airflow cooling the strip. The temperature is significantly decreased and so is the strip tip displacement. The velocity profile in the channel is almost symmetric as the strip has only a small deformation.
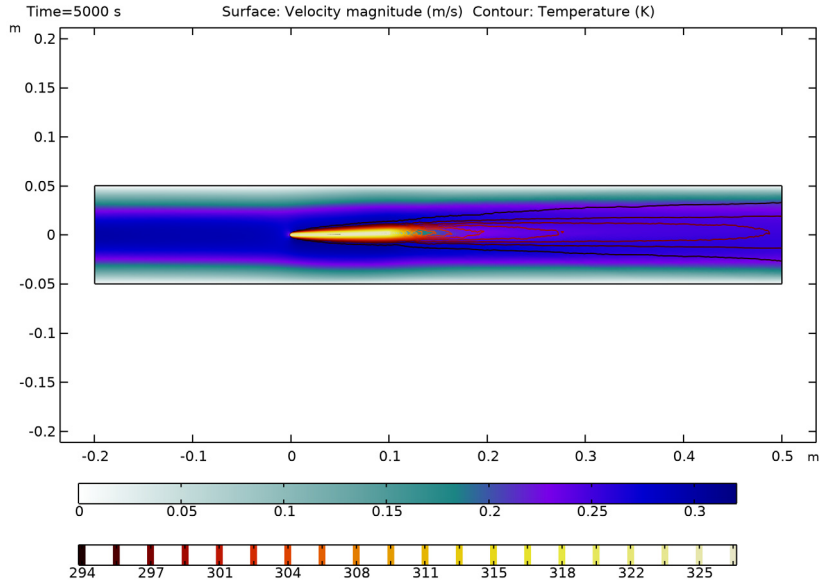
Figure 4: Velocity (surface) and temperature (contour) in the channel at 5000 s.

Figure 5 shows the temperature distribution and the velocity profile in the channel with hot airflow at the final time step. Now the cooling effect is significantly decreased as the air is warm. The strip deformation clearly changes the flow pattern, and most of the air is passing below the structure.
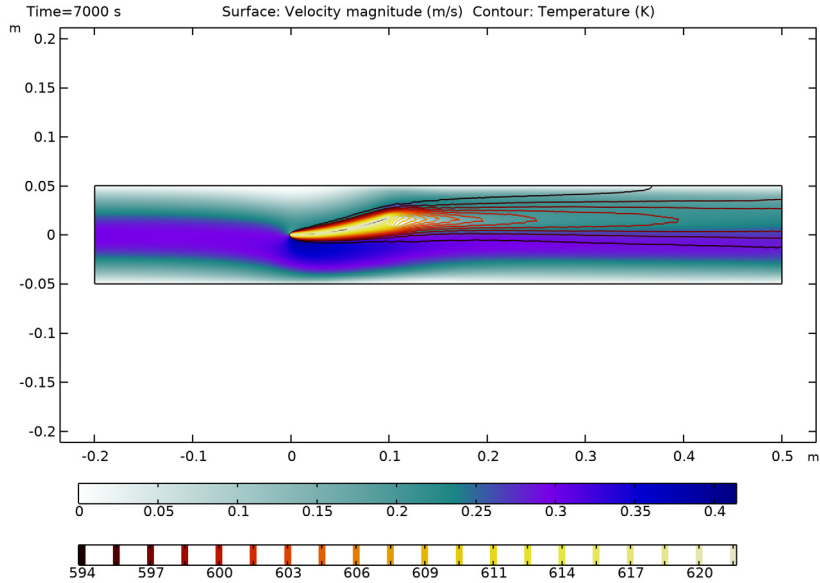
Figure 5: Velocity (surface) and temperature (contour) in the channel at 7000 s.

Notes About the COMSOL Implementation

COMSOL Multiphysics offers the possibility to easily set up this type of combined multiphysics analysis. In the Model Wizard, you can choose the **Fluid-Structure Interaction>Conjugate Heat Transfer>Fluid-Solid Interaction** multiphysics interface and you will automatically get the multiphysics coupling between the fluid flow, the structural deformation, and the heat distribution.

Application Library path: CFD_Module/Fluid-Structure_Interaction/
bimetallic_strip_fsi

Modeling Instructions

From the **File** menu, choose **New**.

In the **New** window, click ⬥ **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click Ω **2D**.

2 In the **Select Physics** tree, select **Fluid Flow>Fluid–Structure Interaction> Conjugate Heat Transfer>Fluid–Solid Interaction**.

3 Click **Add**.

4 Click → **Study**.

5 In the **Select Study** tree, select **General Studies>Time Dependent**.

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
lc	0.7[m]	0.7 m	Length of channel
hc	0.1[m]	0.1 m	Height of channel
ls	0.1[m]	0.1 m	Length of spring
hs	1[mm]	0.001 m	Height of spring

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click ▭ **Rectangle**.

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

3 In the **Width** text field, type ls.

4 In the **Height** text field, type hs.

5 Locate the **Position** section. In the **y** text field, type -hs/2.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	hs/2

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click ▢ **Rectangle**.

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

3 In the **Width** text field, type lc.

4 In the **Height** text field, type hc.

5 Locate the **Position** section. In the **x** text field, type -2*ls.

6 In the **y** text field, type -hc/2.

Form Union (fin)

1 In the **Geometry** toolbar, click ▦ **Build All**.

2 Click the ⟨+⟩ **Zoom Extents** button in the **Graphics** toolbar.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 Select Domain 1 only.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 Select Domains 2 and 3 only.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Fluid 1

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

2 Select Domain 1 only.

MOVING MESH

Deforming Domain 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Moving Mesh** click **Deforming Domain 1**.

2 Select Domain 1 only.

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>Air**.

4 Right-click and choose **Add to Component 1 (comp1)**.

5 In the tree, select **Built-in>Copper**.

6 Right-click and choose **Add to Component 1 (comp1)**.

7 In the tree, select **Built-in>Iron**.

8 Right-click and choose **Add to Component 1 (comp1)**.

9 In the **Home** toolbar, click ⊞ **Add Material** to close the **Add Material** window.

MATERIALS

Copper (mat2)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 Click ⎘ **Paste Selection**.

3 In the **Paste Selection** dialog box, type 2 in the **Selection** text field.

4 Click **OK**.

Iron (mat3)

1 In the **Model Builder** window, click **Iron (mat3)**.

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

3 Click ⎘ **Paste Selection**.

4 In the **Paste Selection** dialog box, type 3 in the **Selection** text field.

5 Click **OK**.

DEFINITIONS

Step 1 (step1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Step**.

2 In the **Settings** window for **Step**, type Vinlet in the **Function name** text field.

3 Locate the **Parameters** section. In the **Location** text field, type 3050[s].

4 In the **To** text field, type 0.2[m/s].

5 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type 100.

LAMINAR FLOW (SPF)

In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

Inlet 1

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Inlet**.

2 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

3 From the list, choose **Fully developed flow**.

4 Locate the **Fully Developed Flow** section. In the U_{av} text field, type `Vinlet(t)`.

5 Select Boundary 1 only.

Outlet 1

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Outlet**.

2 Select Boundary 11 only.

SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

Fixed Constraint 1

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Fixed Constraint**.

2 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.

3 Click 📋 **Paste Selection**.

4 In the **Paste Selection** dialog box, type 4 6 in the **Selection** text field.

5 Click **OK**.

DEFINITIONS

Step 2 (step2)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Step**.

2 In the **Settings** window for **Step**, type `Tinlet` in the **Function name** text field.

3 Locate the **Parameters** section. In the **Location** text field, type `5050[s]`.

4 In the **From** text field, type `20[degC]`.

5 In the **To** text field, type `320[degC]`.

6 Locate the **Smoothing** section. In the **Size of transition zone** text field, type 100.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

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

Inflow 1

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Inflow**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Inflow**, locate the **Upstream Properties** section.

4 In the T_{ustr} text field, type Tinlet(t).

Outflow 1

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Outflow**.

2 Select Boundary 11 only.

Heat Source 1

1 In the **Physics** toolbar, click ⬤ **Domains** and choose **Heat Source**.

2 In the **Settings** window for **Heat Source**, locate the **Domain Selection** section.

3 Click 📋 **Paste Selection**.

4 In the **Paste Selection** dialog box, type 2 3 in the **Selection** text field.

5 Click **OK**.

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

7 In the Q_0 text field, type 50[W]/(ls*hs*ht.dz).

MULTIPHYSICS

Thermal Expansion 1 (te1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Thermal Expansion 1 (te1)**.

2 In the **Settings** window for **Thermal Expansion**, locate the **Domain Selection** section.

3 Click 📋 **Paste Selection**.

4 In the **Paste Selection** dialog box, type 2 3 in the **Selection** text field.

5 Click **OK**.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

Corner Refinement 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Corner Refinement 1** and choose **Build Selected**.

Mapped 1

1 In the **Mesh** toolbar, click ▦ **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

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

4 Select Domains 2 and 3 only.

Distribution 1

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

2 Select Boundaries 4 and 6 only.

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

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

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 8 only.

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

4 In the **Number of elements** text field, type 100.

5 Click ⊞ **Build All**.
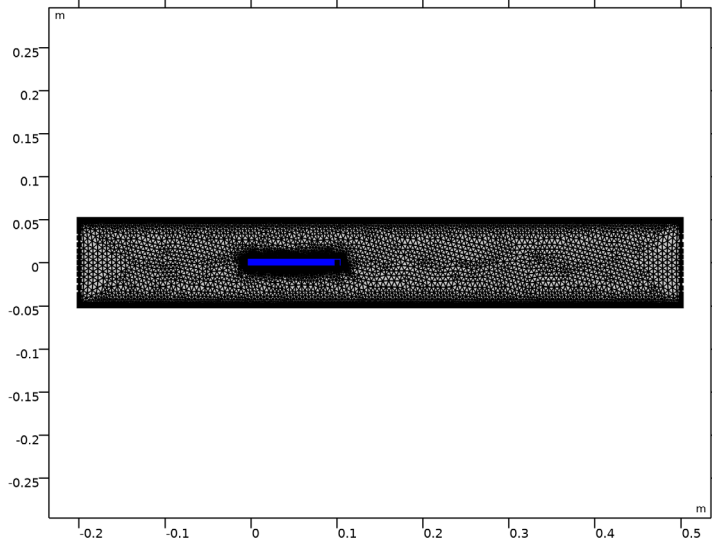
STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type range(0,1e2,7e3).

Solution 1 (sol1)

1 In the **Study** toolbar, click ⊞⊞ **Show Default Solver**.

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

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Dependent Variables 1** node, then click **Pressure (comp1.p)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 From the **Method** list, choose **Manual**.

6 In the **Scale** text field, type 0.1.

7 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Dependent Variables 1** click **Spatial mesh displacement (comp1.spatial.disp)**.

8 In the **Settings** window for **Field**, locate the **Scaling** section.

9 In the **Scale** text field, type 1e-2.

10 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Dependent Variables 1** click **Temperature (comp1.T)**.

11 In the **Settings** window for **Field**, locate the **Scaling** section.

12 From the **Method** list, choose **Manual**.

13 In the **Scale** text field, type 600.

14 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Dependent Variables 1** click **Velocity field (spatial frame) (comp1.u_fluid)**.

15 In the **Settings** window for **Field**, locate the **Scaling** section.

16 From the **Method** list, choose **Manual**.

17 In the **Scale** text field, type 0.5.

18 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Dependent Variables 1** click **Displacement field (comp1.u_solid)**.

19 In the **Settings** window for **Field**, locate the **Scaling** section.

20 In the **Scale** text field, type 1e-2.

21 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Time-Dependent Solver 1**.

22 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

23 From the **Steps taken by solver** list, choose **Intermediate**.

24 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Time-Dependent Solver 1** node, then click **Direct**.

25 In the **Settings** window for **Direct**, locate the **General** section.

26 From the **Solver** list, choose **PARDISO**.

27 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Time-Dependent Solver 1>Segregated 1** node, then click **Displacement Field**.

28 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.

29 From the **Jacobian update** list, choose **On every iteration**.

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

R E S U L T S

Velocity (spf)

1 In the **Settings** window for **2D Plot Group**, locate the **Color Legend** section.

2 From the **Position** list, choose **Bottom**.

3 Click the [+] **Zoom Extents** button in the **Graphics** toolbar.

Surface

1 In the **Model Builder** window, expand the **Velocity (spf)** node, then click **Surface**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Click ❘❙❘ **Change Color Table**.

4 In the **Color Table** dialog box, select **Aurora>AuroraAustralis** in the tree.

5 Click **OK**.

Contour 1

1 In the **Model Builder** window, right-click **Velocity (spf)** and choose **Contour**.

2 In the **Settings** window for **Contour**, locate the **Expression** section.

3 In the **Expression** text field, type T.

4 Locate the **Coloring and Style** section. Click ❘❙❘ **Change Color Table**.

5 In the **Color Table** dialog box, select **Thermal>ThermalDark** in the tree.

6 Click **OK**.

7 In the **Velocity (spf)** toolbar, click ▣ **Plot**.

Pressure (spf)

1 In the **Model Builder** window, under **Results** click **Pressure (spf)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Color Legend** section.

3 From the **Position** list, choose **Bottom**.

4 Click the [+] **Zoom Extents** button in the **Graphics** toolbar.

Stress (solid)

1 In the **Model Builder** window, click **Stress (solid)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Color Legend** section.

3 From the **Position** list, choose **Bottom**.

Surface 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

Temperature (ht)

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

2 In the **Settings** window for **2D Plot Group**, locate the **Plot Settings** section.

3 From the **Frame** list, choose **Spatial (x, y, z)**.

4 Locate the **Color Legend** section. From the **Position** list, choose **Bottom**.

Streamline 1

1 Right-click **Temperature (ht)** and choose **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 1 (comp1)>Laminar Flow> Velocity and pressure>u_fluid,v_fluid - Velocity field (spatial frame)**.

3 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Magnitude controlled**.

4 In the **Temperature (ht)** toolbar, click 🔲 **Plot**.

5 Click the ⊹ **Zoom Extents** button in the **Graphics** toolbar.

Vertical Strip Displacement

1 In the **Home** toolbar, click 📇 **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type Vertical Strip Displacement in the **Label** text field.

Point Graph 1

1 Right-click **Vertical Strip Displacement** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Selection** section.

3 Click 📋 **Paste Selection**.

4 In the **Paste Selection** dialog box, type 7 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics>Displacement>Displacement field - m>v_solid - Displacement field, Y-component**.

7 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

8 In the **Vertical Strip Displacement** toolbar, click 🔲 **Plot**.