

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.



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.

NEW

In the New window, click 🔗 Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 🧐 2D.
- 2 In the Select Physics tree, select Fluid Flow>Fluid-Structure Interaction> Conjugate Heat Transfer>Fluid-Solid Interaction.
- 3 Click Add.
- 4 Click \bigcirc Study.
- 5 In the Select Study tree, select General Studies>Time Dependent.
- 6 Click **M** Done.

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.

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

3 In the table, enter the following settings:

GEOMETRY I

Rectangle 1 (r1)

- I 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 1s.
- 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)

I 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 1c.
- 4 In the **Height** text field, type hc.
- 5 Locate the Position section. In the x text field, type -2*1s.
- 6 In the y text field, type -hc/2.

Form Union (fin)

I In the Geometry toolbar, click 🟢 Build All.

2 Click the **Comextents** button in the **Graphics** toolbar.

LAMINAR FLOW (SPF)

I In the Model Builder window, under Component I (compl) click Laminar Flow (spf).

2 Select Domain 1 only.

SOLID MECHANICS (SOLID)

I In the Model Builder window, under Component I (compl) click Solid Mechanics (solid).

2 Select Domains 2 and 3 only.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Fluid I

- I In the Model Builder window, under Component I (compl)> Heat Transfer in Solids and Fluids (ht) click Fluid I.
- 2 Select Domain 1 only.

MOVING MESH

Deforming Domain I

- I In the Model Builder window, under Component I (compl)>Moving Mesh click Deforming Domain I.
- 2 Select Domain 1 only.

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>Air.
- 4 Right-click and choose Add to Component I (compl).
- 5 In the tree, select Built-in>Copper.
- 6 Right-click and choose Add to Component I (compl).
- 7 In the tree, select Built-in>Iron.
- 8 Right-click and choose Add to Component I (compl).
- 9 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

MATERIALS

Copper (mat2)

- I 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)

- I 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 | (step |)

- I 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 I (compl) click Laminar Flow (spf).

Inlet 1

- I 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 I

- I In the Physics toolbar, click Boundaries and choose Outlet.
- **2** Select Boundary 11 only.

SOLID MECHANICS (SOLID)

In the Model Builder window, under Component I (compl) click Solid Mechanics (solid).

Fixed Constraint I

- I 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 (step 2)

- I 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 I (compl) click Heat Transfer in Solids and Fluids (ht).

Inflow I

- I 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 I

- I In the Physics toolbar, click Boundaries and choose Outflow.
- **2** Select Boundary 11 only.

Heat Source 1

- I 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 (tel)

- I In the Model Builder window, under Component I (compl)>Multiphysics click Thermal Expansion I (tel).
- 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 I

- I In the Model Builder window, under Component I (comp1) right-click Mesh I 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 I

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

Mapped I

- I In the Mesh toolbar, click III 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 I

- I Right-click Mapped I and choose Distribution.
- **2** Click the **Com Box** button in the **Graphics** toolbar.
- **3** Select Boundaries 4 and 6 only.
- 4 In the Settings window for Distribution, locate the Distribution section.
- 5 In the Number of elements text field, type 2.

Distribution 2

- I In the Model Builder window, right-click Mapped I 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.



STUDY I

Step 1: Time Dependent

- I In the Model Builder window, under Study I click Step I: 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 (soll)

- I In the Study toolbar, click The 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 (soll)>Dependent Variables I node, then click Pressure (compl.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 I>Solver Configurations>Solution I (soll)> Dependent Variables I click Spatial mesh displacement (compl.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 I>Solver Configurations>Solution I (soll)> Dependent Variables I click Temperature (compl.T).
- II In the Settings window for Field, locate the Scaling section.
- 12 From the Method list, choose Manual.
- **I3** In the **Scale** text field, type 600.
- I4 In the Model Builder window, under Study I>Solver Configurations>Solution I (soll)> Dependent Variables I click Velocity field (spatial frame) (compl.u_fluid).
- 15 In the Settings window for Field, locate the Scaling section.
- **I6** From the **Method** list, choose **Manual**.
- **I7** In the **Scale** text field, type 0.5.
- 18 In the Model Builder window, under Study I>Solver Configurations>Solution I (solI)> Dependent Variables I click Displacement field (compl.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 I>Solver Configurations>Solution I (soll) click Time-Dependent Solver I.
- **22** In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
- **2** From the Steps taken by solver list, choose Intermediate.
- 24 In the Model Builder window, expand the Study I>Solver Configurations> Solution I (soll)>Time-Dependent Solver I node, then click Direct.
- 25 In the Settings window for Direct, locate the General section.
- **26** From the **Solver** list, choose **PARDISO**.
- In the Model Builder window, expand the Study I>Solver Configurations>
 Solution I (soll)>Time-Dependent Solver I>Segregated I 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**.

RESULTS

Velocity (spf)

- I In the Settings window for 2D Plot Group, locate the Color Legend section.
- 2 From the **Position** list, choose **Bottom**.
- **3** Click the 4 **Zoom Extents** button in the **Graphics** toolbar.

Surface

- I 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 I

- I 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)

- I 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 4 **Zoom Extents** button in the **Graphics** toolbar.

Stress (solid)

- I 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

- I In the Model Builder window, expand the Stress (solid) node, then click Surface I.
- 2 In the Settings window for Surface, locate the Expression section.

3 From the Unit list, choose MPa.

Temperature (ht)

- I 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

- I 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 I (compl)>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 **I** Plot.

Isothermal Contours (ht)

- I In the Model Builder window, under Results click Isothermal Contours (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.
- **5** Click the **i Zoom Extents** button in the **Graphics** toolbar.

Vertical Strip Displacement

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Vertical Strip Displacement in the Label text field.

Point Graph I

- I 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 I (compl)>

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 **O** Plot.