Created in COMSOL Multiphysics 6.0

Dynamic Wall Heat Exchanger

Introduction

The dynamic wall heat exchanger aims to improve heat transfer in small scale heat exchangers and to reduce pressure drop by using a dynamic wall with an oscillating wave shape. The model is inspired from Ref. 1.

Such deformations of the fluid domain induce mixing of the fluid and reduce the formation of thermal boundary layers. Hence, they increase heat transfer between the walls and the fluid. In addition, the wave shaped deformation induces a pumping effect similar to a peristaltic pumping which compensates the pressure losses.

Model Definition

Figure 1 shows the geometry of the heat flux in a static configuration with a flat upper wall. Water is heated as it flows from the inlet to the outlet due to a heat flux at the bottom side of the heat sink. The delivered heat rate at the bottom boundary is set to 125 W.

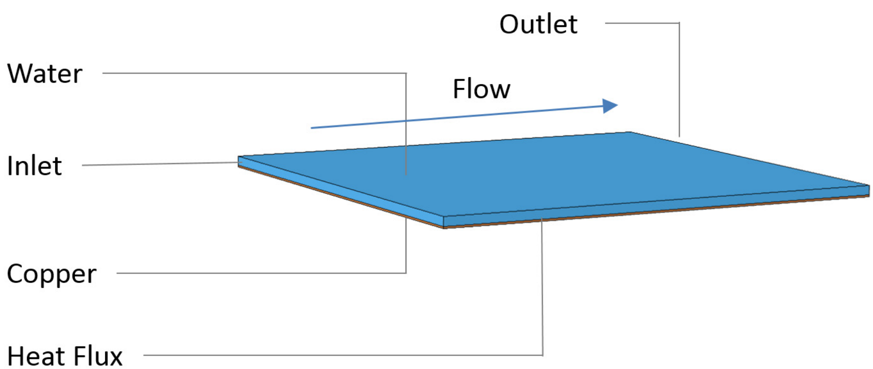

Figure 1: Geometry and operating conditions of the heat exchanger.

The upper wall of the dynamic wall heat exchanger is deformed to obtained a transient oscillating shape. The upper wall displacement, u, is defined by

$$u = AYf_X(X)\sin\left(2\pi\left(-tf+X\frac{N}{L}\right)\right)f_t(t) \qquad (1)$$

where:

- X and Y are the space coordinates on the material frame
- t is the time variable
- A is the ratio of oscillation amplitude to channel height

- $f_X(X)$ gives the normalized amplitude of the spatial oscillations in the X direction
- $f_t(t)$ is the normalized amplitude of the oscillations as a function of time and included a smooth starting effect
- N is the number of waves in the channel length direction
- L is the channel length
- f is the oscillation frequency

Figure 2 shows the shape of the oscillating wall in a cross section parallel to the flow direction after the starting effect.

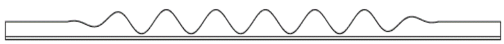

Figure 2: Oscillating shape of the wall heat exchanger.

Results

In a first step, the performances of the heat exchanger are evaluated for a static configuration. A stationary analysis is computed with no displacement of the upper channel wall.

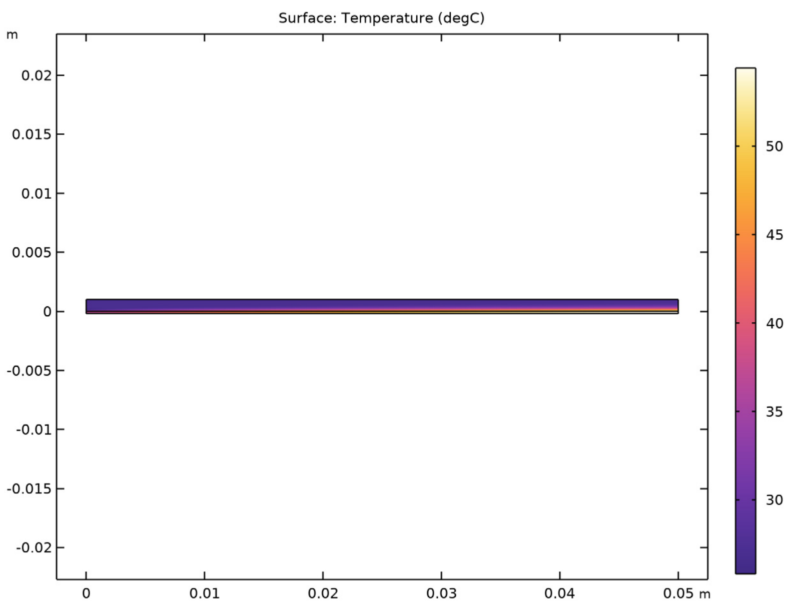

Figure 3: Temperature distribution in the static heat exchanger.

Figure 3 shows the temperature distribution in the channel in stationary regime where the mass flow rate is 5.5 g/s.

The overall heat transfer coefficient of the heat exchanger is defined as

$$ = \frac{P_0}{\overline{T}_{\mathrm{wall}} - \overline{T}_{\mathrm{channel}}}. \qquad (2)$$

and evaluates to about 2,900 W/(m²·K) in this application.

Then, a time dependent analysis is performed where the upper wall displacement is prescribed according to Equation 2, using an oscillation amplitude of 90%. The average outlet temperature and the mass flow rate is monitored during the simulation using probes. As shown in Figure 4, a pseudo-periodic regime is reached after 0.6 s.

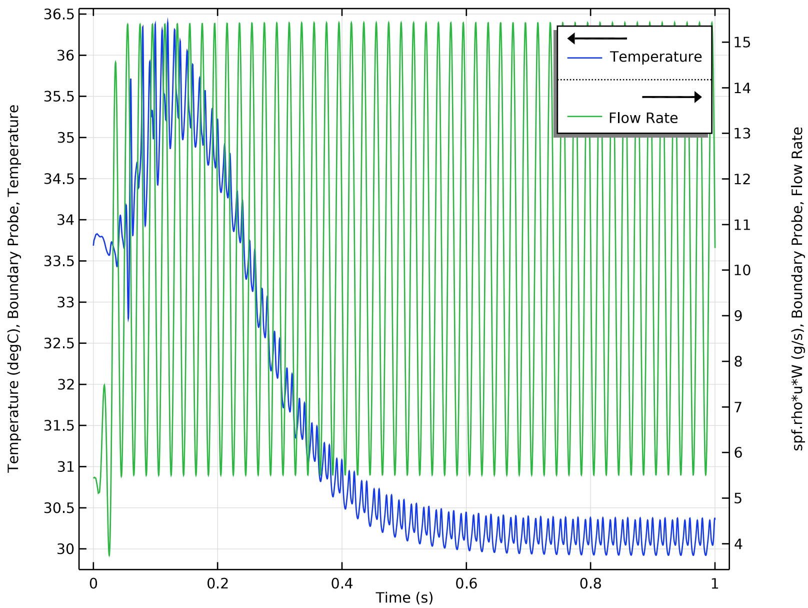

Figure 4: Flow rate and outlet temperature variations during the simulation.

As expected, the flow rate increases compared to the static case. The average value over the last period is 10.5 g/s, which is close to twice higher than in the static case.

Figure 5 shows the temperature profile in the dynamic wall heat exchanger at time 0.6 s.

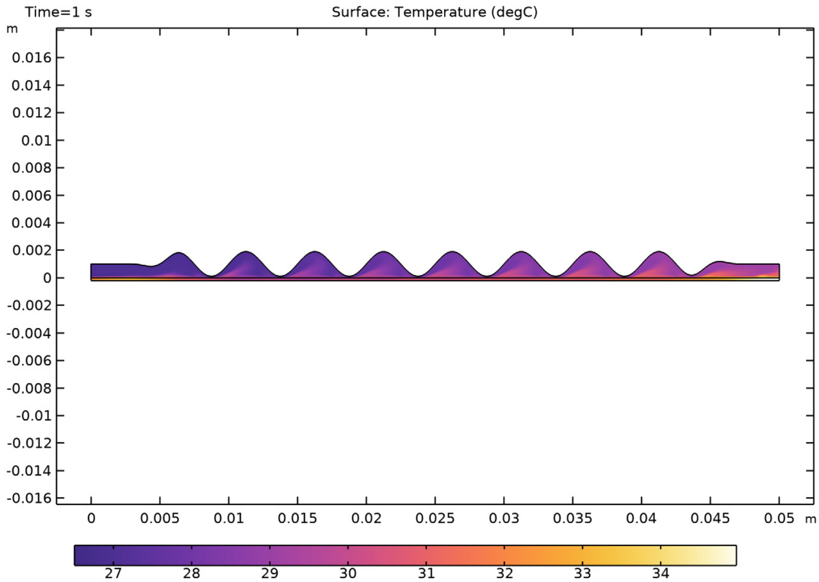

Figure 5: Temperature distribution in the dynamic wall heat exchanger.

Finally, the global heat transfer coefficient is estimated using a time average over the last period

$$\langle h \rangle = \left\langle \frac{P_0}{\bar{T}_{\text{wall}} - \bar{T}_{\text{channel}}} \right\rangle \tag{3}$$

Again, the use of a dynamic wall improves the overall heat rate, equal to 19,000 W/(m^2·K) with an oscillation amplitude of 90%.

Reference

1. P. Kumar, K. Schmidmayer, F. Topin, and M. Miscevic, “Heat transfer enhancement by dynamic corrugated heat exchanger wall: Numerical study,” *Journal of Physics: Conference Series*, vol. 745, 2016.

Application Library path: `Heat_Transfer_Module/Heat_Exchangers/dynamic_wall_heat_exchanger`

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 **2D**.
2 In the **Select Physics** tree, select **Heat Transfer>Conjugate Heat Transfer>Laminar Flow**.
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 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `dynamic_wall_heat_exchanger_parameters.txt`.

GEOMETRY 1

The first domain corresponds to the water channel.

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 `L`.
4 In the **Height** text field, type `H`.

The second domain corresponds to the copper wall.

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 `L`.
4. In the **Height** text field, type `H/5`.
5. Locate the **Position** section. In the **y** text field, type `-H/5`.

DEFINITIONS

Linear Projection 1 (linproj1)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Linear Projection**.
2. Select Domain 2 only.
3. In the **Settings** window for **Linear Projection**, locate the **Source Vertices** section.
4. Click to select the **Activate Selection** toggle button for **Source vertex 1**.
5. Select Point 2 only.
6. Click to select the **Activate Selection** toggle button for **Source vertex 2**.
7. Select Point 5 only.
8. Click to select the **Activate Selection** toggle button for **Source vertex 3**.
9. Select Point 3 only.
10. Locate the **Destination Vertices** section. Click to select the **Activate Selection** toggle button for **Destination vertex 1**.
11. Select Point 2 only.
12. Click to select the **Activate Selection** toggle button for **Destination vertex 2**.
13. Select Point 5 only.

Average 1 (aveop1)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.
2. In the **Settings** window for **Average**, locate the **Source Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundary 4 only.

Next, import variables that are needed to define the channel deformation and for postprocessing purposes.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `dynamic_wall_heat_exchanger_variables.txt`.

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>Water, liquid**.

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

MATERIALS

Copper (mat2)

Select Domain 1 only.

DEFINITIONS

Ambient Properties 1 (ampr1)

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

2 In the **Settings** window for **Ambient Properties**, locate the **Ambient Conditions** section.

3 In the T_{amb} text field, type `T0`.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Now, define the settings of the Heat Transfer interface.

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

2 In the **Settings** window for **Heat Transfer in Solids and Fluids**, locate the **Physical Model** section.

3 In the d_z text field, type `W`.

4 In the T_{ref} text field, type `T0`.

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 2 only.

Initial Values 1

1. In the **Model Builder** window, 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)**.

Inflow 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.
2. Select Boundary 3 only.
3. In the **Settings** window for **Inflow**, locate the **Upstream Properties** section.
4. From the T_{ustr} list, choose **Ambient temperature (ampr1)**.
5. Locate the **Boundary Selection** section. Click **Create Selection**.
6. In the **Create Selection** dialog box, type `Inlet` in the **Selection name** text field.
7. Click **OK**.

Outflow 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Outflow**.
2. Select Boundary 7 only.
3. In the **Settings** window for **Outflow**, locate the **Boundary Selection** section.
4. Click **Create Selection**.
5. In the **Create Selection** dialog box, type `Outlet` in the **Selection name** text field.
6. Click **OK**.

Heat Flux 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2. Select Boundary 2 only.
3. In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
4. From the **Flux type** list, choose **Heat rate**.
5. In the P_0 text field, type `P0`.

LAMINAR FLOW (SPF)

The following instructions set the **Laminar Flow** interface active on the water domain only, and models the flow as incompressible by using a constant density in the simulation. The reference temperature is set in the Heat Transfer interface.

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

2 Select Domain 2 only.

3 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.

4 From the **Compressibility** list, choose **Incompressible flow**.

Inlet 1

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

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

3 From the **Selection** list, choose **Inlet**.

4 Locate the **Boundary Condition** section. From the list, choose **Pressure**.

5 Locate the **Pressure Conditions** section. In the p_0 text field, type `Pin`.

Outlet 1

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

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

3 From the **Selection** list, choose **Outlet**.

MESH 1

Mapped 1

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

Size

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Calibrate for** list, choose **Fluid dynamics**.

4 From the **Predefined** list, choose **Extra coarse**.

Now, compute the stationary solution for the static configuration of the heat exchanger.

5 In the **Home** toolbar, click **Compute**.

RESULTS

Surface

The first default plot shows the temperature profile in the channel (see Figure 3).

1 In the **Model Builder** window, expand the **Results>Temperature (ht)** node, then click **Surface**.

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

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

Mass Flow Rate, Static Case

1 In the **Results** toolbar, click **More Derived Values** and choose **Integration>Line Integration**.

2 In the **Settings** window for **Line Integration**, type `Mass Flow Rate, Static Case` in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **Outlet**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`spf.rho*u*W`	`g/s`	`Mass flow rate`

5 Click **Evaluate**.

Heat Transfer Coefficient, Static Case

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Heat Transfer Coefficient, Static Case` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>htc - Global heat transfer coefficient - W/(m²·K)**.

4 Click **Evaluate**.

DEFINITIONS

Now, update the model to define the dynamic wall.

Boundary Probe, Temperature

1 In the **Definitions** toolbar, click **Probes** and choose **Boundary Probe**.

2 In the **Settings** window for **Boundary Probe**, type `Boundary Probe, Temperature` in the **Label** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **Outlet**.

4 Locate the **Expression** section. From the **Table and plot unit** list, choose **degC**.

Boundary Probe, Flow Rate

1 Right-click **Boundary Probe, Temperature** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Probe**, type `Boundary Probe, Flow Rate` in the **Label** text field.

3 Locate the **Probe Type** section. From the **Type** list, choose **Integral**.

4 Locate the **Expression** section. In the **Expression** text field, type `spf.rho*u*W`.

5 In the **Table and plot unit** field, type `g/s`.

6 In the **Definitions** toolbar, click **Moving Mesh** and choose **Domains> Prescribed Deformation**.

MOVING MESH

Prescribed Deformation 1

1 Select Domain 2 only.

2 In the **Settings** window for **Prescribed Deformation**, locate the **Prescribed Deformation** section.

3 Specify the *dx* vector as

0	X
disp	Y

The **Moving Mesh** subtree, containing **Prescribed Deformation**, is only relevant for time-dependent studies. In the next steps, disable **Moving Mesh** from **Study 1** as this is a stationary study, and create a new time-dependent study accounting for moving mesh.

STUDY 1

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (Comp1)>Moving Mesh**.

5 Click **Control Frame Deformation**.

6 In the tree, select **Component 1 (Comp1)>Moving Mesh**.

7 Click **Disable in Solvers**.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Time Dependent

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

2 In the **Output times** text field, type `range(0,step,end)`.

3 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

4 From the **Method** list, choose **Solution**.

5 From the **Study** list, choose **Study 1, Stationary**.

Prescribe a maximum time step to the solver.

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.

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

4 From the **Maximum step constraint** list, choose **Constant**.

5 In the **Maximum step** text field, type `1/(5*freq)`.

6 In the **Study** toolbar, click **Compute**.

During the computation, you can visualize how the mass flow rate and average outlet temperature vary in time with the displayed probe plots. Notice that after a number of periods the solution becomes identical over the periods, which indicates that the system has reached a pseudo-stationary state.

RESULTS

Temperature (ht) 1

The first default plot shows the temperature profile in the channel (see Figure 5).

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

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

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

Surface

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

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

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

4 In the **Temperature (ht) 1** toolbar, click **Plot**.

The probe plot can also be displayed by selecting the corresponding node (see Figure 4).

Temperature and Flow Rate

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 9**.

2 In the **Settings** window for **1D Plot Group**, type `Temperature and Flow Rate` in the **Label** text field.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Temperature and Flow Rate** node, then click **Probe Table Graph 1**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 In the **Columns** list, select **Temperature (degC), Boundary Probe, Temperature**.

4 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
`Temperature`

6 Click to select row number 2 in the table.

7 Click **Delete**.

Table Graph 2

1 In the **Model Builder** window, right-click **Temperature and Flow Rate** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.
3 From the **Table** list, choose **Probe Table 3**.
4 From the **Plot columns** list, choose **Manual**.
5 In the **Columns** list, select **spf.rho*u*W (g/s), Boundary Probe, Flow Rate**.
6 Locate the **Legends** section. Select the **Show legends** check box.
7 From the **Legends** list, choose **Manual**.
8 In the table, enter the following settings:

Legends
`Flow Rate`

Temperature and Flow Rate

1 In the **Model Builder** window, click **Temperature and Flow Rate**.
2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.
3 Select the **Two y-axes** check box.
4 In the table, select the **Plot on secondary y-axis** check box for **Table Graph 2**.
5 In the **Temperature and Flow Rate** toolbar, click **Plot**.

Mass Flow Rate, Static Case

Duplicate the two derived values defined in the first study and update them to compute the average over the last period of the flow rate and of the global heat transfer coefficient.

Mass Flow Rate, Dynamic Case

1 In the **Model Builder** window, right-click **Mass Flow Rate, Static Case** and choose **Duplicate**.
2 In the **Settings** window for **Line Integration**, type `Mass Flow Rate, Dynamic Case` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
4 From the **Time selection** list, choose **Interpolated**.
5 In the **Times (s)** text field, type `range(end-1/freq,step,end)`.
6 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Average**.
7 Click **Evaluate**.

Heat Transfer Coefficient, Dynamic Case

1 In the **Model Builder** window, right-click **Heat Transfer Coefficient, Static Case** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, type `Heat Transfer Coefficient, Dynamic Case` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 From the **Time selection** list, choose **Interpolated**.

5 In the **Times (s)** text field, type `range(end-1/freq,step,end)`.

6 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Average**.

To create a 3D plot showing both the temperature at the bottom of the heat sink and the fluid velocity, follow the instructions below.

7 Click **Evaluate**.

Extrusion 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Extrusion 2D**.

2 In the **Settings** window for **Extrusion 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Extrusion** section. In the **z maximum** text field, type `0.01`.

Velocity and Temperature 3D

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Velocity and Temperature 3D` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Velocity

1 Right-click **Velocity and Temperature 3D** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Velocity` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `spf.U`.

4 In the **Velocity and Temperature 3D** toolbar, click **Plot**.

Temperature

1 In the **Model Builder** window, right-click **Velocity and Temperature 3D** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Temperature` in the **Label** text field.

3 Locate the **Expression** section. From the **Unit** list, choose **degC**.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **HeatCameraLight**.

Translation 1

1 Right-click **Temperature** and choose **Translation**.

2 In the **Settings** window for **Translation**, locate the **Translation** section.

3 In the **x** text field, type `0`.

4 In the **y** text field, type `-0.005`.

5 In the **z** text field, type `0`.

6 In the **Velocity and Temperature 3D** toolbar, click **Plot**.

Temperature (ht) 1

Finally, create an animation showing the shape of the channel and the temperature distribution over time.

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

Animation 1

1 In the **Temperature (ht) 1** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Frames** section.

3 From the **Frame selection** list, choose **All**.

4 Click the **Play** button in the **Graphics** toolbar.