Model created in COMSOL Multiphysics 6.4

Nonisothermal Turbulent Flow over a Flat Plate

Introduction

This model of turbulent airflow over a flat plate validates the skin friction coefficient against the White's correlation and compares the simulated heat transfer coefficient with Nusselt number based correlations. The simulation results are in good agreement with empirical correlations.

Model Definition

A coupled heat transfer and airflow problem is solved using the Nonisothermal Flow multiphysics interface in a 2D geometry:

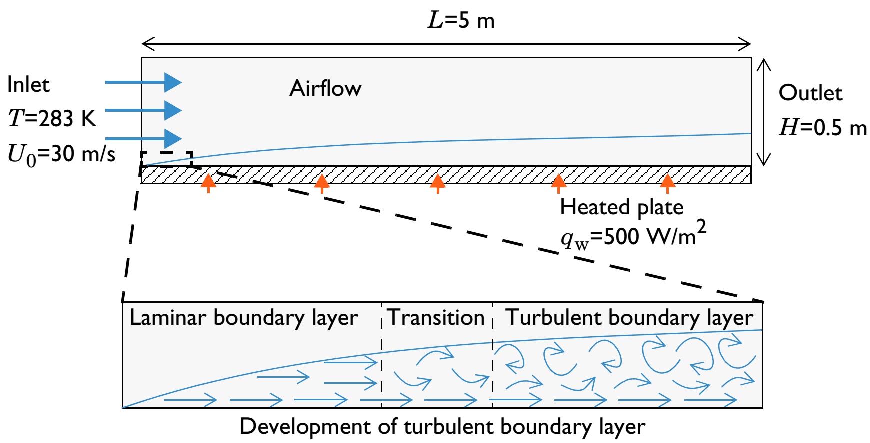

Figure 1: Schematic view of the model definition. Note that the boundary layer thickness is exaggerated for clarity.

A laminar airflow with uniform velocity profile U_0 = 30 m/s and uniform temperature profile T_0 = 283 K enters the domain through the left boundary. It flows above a plate of length L heated with a constant heat flux q_w of 500 W/m^2. Turbulence quickly develops in the boundary layer due to the high velocity of the air.

TURBULENCE MODELING AND WALL TREATMENT

The turbulent airflow is modeled by the Reynolds-averaged Navier–Stokes (RANS) equations, by using the Turbulent Flow, SST version of the Nonisothermal Flow multiphysics interface. The **Automatic** option for **Wall treatment** provided by this interface allows using wall functions when the boundary layer mesh is coarse, and to switch to a low Reynolds number formulation when the mesh is fine enough in the boundary layer.

Because of the high velocity of the airflow, the laminar and transition boundary layers can be neglected. In a setup where the laminar boundary layer is of importance and later switches to a turbulent boundary layer, the SST turbulence model proposes an option to model transition. The **Include transition modeling** option is available in the Turbulent Flow, SST physics node.

The **Inlet** boundary condition is used to set the laminar inlet at the left boundary of the computational domain by imposing both a uniform velocity profile and no turbulent intensity.

NUSSELT NUMBER AND SKIN FRICTION CORRELATIONS

In Ref. 1 (Eq. 6), the Nusselt number correlation is given for a turbulent flow over a plate either at uniform temperature, or heated with a uniform heat flux. At the position x along the heated plate, the Nusselt number Nu_x is given by the following formula:

$$\mathrm{Nu}_x = \frac{\mathrm{Re}_x \mathrm{Pr} \frac{C_\mathrm{f}}{2}}{1 + 12{,}7(\mathrm{Pr}^{2/3} - 1)\sqrt{\frac{C_\mathrm{f}}{2}}},$$

where Re_x is the Reynolds number at the position x along the heated plate and at film temperature $T_{\mathrm{f},x}$, defined by

$$\mathrm{Re}_x = \frac{\rho(T_{\mathrm{f},x})U_0 x}{\mu(T_{\mathrm{f},x})}.$$

Pr is the Prandtl number at film temperature $T_{\mathrm{f},x}$, defined by

$$\mathrm{Pr} = \frac{C_p(T_{\mathrm{f},x})\mu(T_{\mathrm{f},x})}{k(T_{\mathrm{f},x})}.$$

C_f is the skin friction coefficient, defined for example by White's correlation:

$$C_\mathrm{f} = \frac{0{,}455}{(\ln(0{,}06\mathrm{Re}_x))^2},$$

where $\rho(T_{\mathrm{f},x})$ (SI unit: kg/m^3) denotes the density, $\mu(T_{\mathrm{f},x})$ (SI unit: Pa·s) the dynamic viscosity, $C_p(T_{\mathrm{f},x})$ (SI unit: J/(kg·K)) the heat capacity at constant pressure, and $k_\mathrm{f}(T_{\mathrm{f},x})$ (SI unit: W/(m·K)) the thermal conductivity, and $T_{\mathrm{f},x} = (T_0 + T_{\mathrm{w},x})/2$, with $T_{\mathrm{w},x}$ the plate surface temperature.

The heat transfer coefficient, h (SI unit: $W/(m^2{\cdot}K)$), at the surface of the heated plate is then expressed as

$$h = \frac{k\mathrm{Nu}_x}{x}.$$

To validate the results of the numerical simulation, the skin friction coefficient and the heat transfer coefficient obtained from the above correlations are compared with the values calculated from the computed velocity and temperature fields, as follows

$$C_\mathrm{f} = \frac{2\tau_w}{\rho(T_{\mathrm{f},x})U_0^2},$$

$$h = \frac{q_\mathrm{w}}{T_{\mathrm{w},x} - T_{\mathrm{b},x}},$$

where τ_w is a shear wall stress

$$\tau_w = \frac{\rho u_\tau^2}{2},$$

and $T_{\mathrm{b},x}$ is the bulk temperature at the position x along the heated plate

$$T_{\mathrm{b},x} = \frac{\int_0^b u(x,y)T(x,y)dy}{\int_0^b u(x,y)dy}.$$

Results and Discussion

A numerical convergence study based on the mesh refinement is run using the `mesh_coeff` parameter. The parameter is chosen so that the first fluid cell falls in the turbulent sublayer ($y^+ > 30$), in the buffer layer ($1 < y^+ < 30$), and in the laminar sublayer ($y^+ < 1$) as shown in Figure 2. This way, results are compared when the **Automatic** wall treatment operates in wall function mode, low Reynolds mode, and the blending of the two.

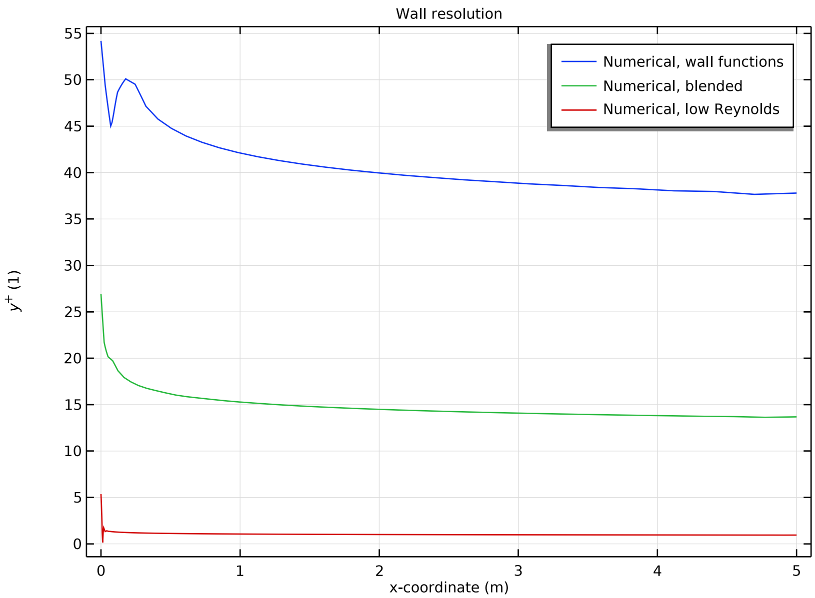

Figure 2: Wall resolution in viscous units (y^+).

The comparison of the computed skin friction coefficient with that estimated using White's correlation (Figure 3) shows good agreement across the plate for all three approaches (wall functions, low Reynolds, and blended).

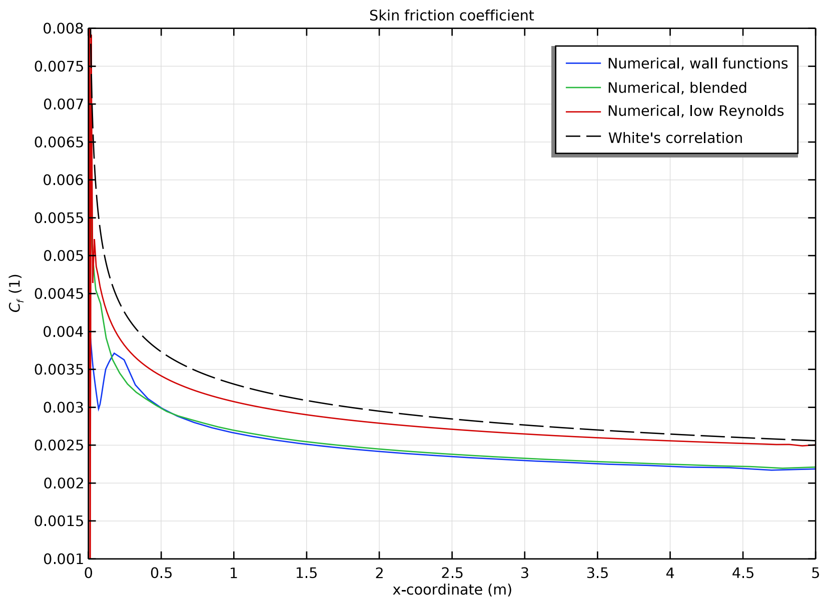

Figure 3: Comparison of the computed skin friction coefficient with values estimated using White's correlation.

A similar level of agreement is observed in the comparison of the heat transfer coefficient with values obtained from the Nusselt number correlations (Figure 4).

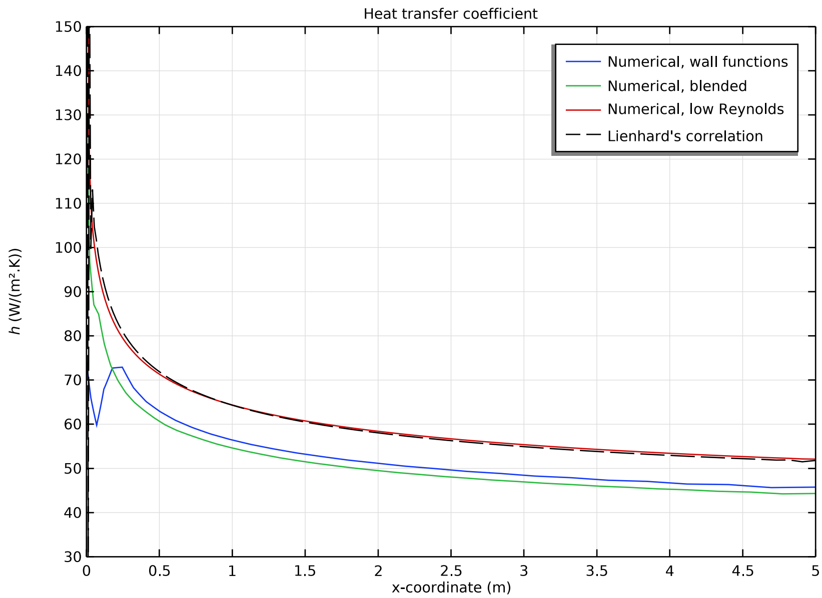

Figure 4: Comparison of the computed heat transfer coefficient with the heat transfer coefficient estimations based on Nusselt number correlations.

References

1. J.H. Lienhard V, *Heat transfer in flat-plate boundary layers: a correlation for laminar, transitional, and turbulent flow*, Journal of Heat Transfer, 2020.

Application Library path: `Heat_Transfer_Module/Verification_Examples/flat_plate_nitf_turbulent`

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 **Fluid Flow** > **Nonisothermal Flow** > **Turbulent Flow** > **Turbulent Flow, SST**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics** > **Stationary with Initialization**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

First, define parameters for the geometry, the inlet conditions, and the heat flux applied on the plate.

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
L	5[m]	5 m	Plate length
b	0.5[m]	0.5 m	Height
T0	283[K]	283 K	Inlet temperature
U0	30[m/s]	30 m/s	Inlet velocity
qw	500[W/m^2]	500 W/m²	Wall heat flux
mesh_coeff	3	3	Mesh coefficient for parametric study

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 L.
4 In the **Height** text field, type b.
5 Click **Build All Objects**.

ADD MATERIAL

1 In the **Materials** 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 Click the **Add to Component** button in the window toolbar.
5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air (mat1)

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

DEFINITIONS

Variables 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.
2 Right-click **Definitions** and choose **Variables**.

   Define the material properties of the airflow at film conditions for the computation of the Nusselt correlation.

3 In the **Settings** window for **Variables**, locate the **Variables** section.
4 In the table, enter the following settings:

Name	Expression	Unit	Description
Tb	integrate(comp1.at2(x,y,u*T),y,0,b)/integrate(comp1.at2(x,y,u),y,0,b)	K	Bulk temperature
T_film	0.5*(T+T0)	K	Film temperature
rho_film	mat1.def.rho(ht.pA,T_film)	kg/m³	Film density
k_film	mat1.def.k(T_film)	W/(m·K)	Film thermal conductivity
Cp_film	mat1.def.Cp(T_film)	J/(kg·K)	Film heat capacity
mu_film	mat1.def.eta(T_film)	Pa·s	Film viscosity

Name	Expression	Unit	Description
Pr_film	Cp_film*mu_film/k_film		Prandtl number based on film properties
Re_film	rho_film*U0*x/mu_film		Reynolds number based on film properties
Cf_film	(nitf1.rho*spf.u_tauWall^2)/(0.5*rho_film*U0^2)		Skin friction coefficient based on film properties
Cf_White	0.455/((log(0.06*max(Re_film,1)))^2)		Skin friction coefficient (White)
Nu_x_turb_Lienhard	Re_film*Pr_film*(Cf_film/2)/(1+12.7*(Pr_film^(2/3)-1)*sqrt(Cf_film/2))		Nusselt number (Lienhard)

TURBULENT FLOW, SST (SPF)

Set the domain and boundary conditions for the definition of the compressible airflow. An **Automatic** wall treatment is set by default in the turbulence model.

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

2 In the **Settings** window for **Turbulent Flow, SST**, locate the **Physical Model** section.

3 From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Turbulent Flow, SST (spf)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 Specify the **u** vector as

U0	x
0	y

Inlet 1

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

2 Select Boundary 1 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type `U0`.

5 Locate the **Turbulence Conditions** section. From the I_T list, choose **User defined**.

6 In the text field, type `0`.

Outlet 1

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

2 Select Boundary 4 only.

A symmetry boundary condition is applied at the top of the domain to improve numerical convergence.

Symmetry 1

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

2 Select Boundary 3 only.

HEAT TRANSFER IN FLUIDS (HT)

Set the domain and boundary conditions for the definition of heat transfer in air over the heated plate.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in 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 `T0`.

Outflow 1

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

2 Select Boundary 4 only.

Symmetry 1

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

2 Select Boundary 3 only.

Heat Flux 1

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

2 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

3 In the q_0 text field, type `qw`.

4 Select Boundary 2 only.

MESH 1

Set manually a mapped mesh for the numerical convergence study, with refinement in the boundary layer over the plate.

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

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

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

Size 1

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

2 Click **Yes** to confirm.

Corner Refinement 1

1 In the **Model Builder** window, right-click **Corner Refinement 1** and choose **Delete**.

2 Click **Yes** to confirm.

Free Triangular 1

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Delete**.

2 Click **Yes** to confirm.

Boundary Layers 1

1 In the **Model Builder** window, right-click **Boundary Layers 1** and choose **Delete**.

2 Click **Yes** to confirm.

Mapped 1

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

Distribution (horizontal)

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

2 In the **Settings** window for **Distribution**, type `Distribution (horizontal)` in the **Label** text field.

3 Select Boundaries 2 and 3 only.

4 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `L*2*mesh_coeff`.

6 In the **Element ratio** text field, type `10`.

7 Select the **Reverse direction** checkbox.

Distribution (vertical)

1 Right-click **Distribution (horizontal)** and choose **Duplicate**.

2 In the **Settings** window for **Distribution**, type `Distribution (vertical)` in the **Label** text field.

3 Locate the **Boundary Selection** section. Click **Clear Selection**.

4 Select Boundaries 1 and 4 only.

5 Locate the **Distribution** section. In the **Number of elements** text field, type `10*mesh_coeff`.

6 From the **Growth rate** list, choose **Exponential**.

7 In the **Element ratio** text field, type `15*mesh_coeff*(mesh_coeff-1)-30`.

8 Clear the **Reverse direction** checkbox.

9 Click **Build All**.

MESH 1

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Mesh 1** node.

STUDY 1

Add a parametric sweep for the numerical convergence study.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
mesh_coeff (Mesh coefficient for parametric study)	3 4 10	

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

RESULTS

The default plot groups show the distributions of velocity, pressure, temperature, and the wall resolution. Follow the instructions below to reproduce the plots shown in Figure 2.

Wall y+

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

2 In the **Settings** window for **1D Plot Group**, type `Wall y+` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol3)**.

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

5 In the **Title** text area, type `Wall resolution`.

6 Locate the **Plot Settings** section.

7 Select the **y-axis label** checkbox. In the associated text field, type `\$y^+\$ (1)`.

8 Click the **Zoom Extents** button in the **Graphics** toolbar.

Numerical

1 Right-click **Wall y+** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, type `Numerical` in the **Label** text field.

3 Select Boundary 2 only.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `spf.Delta_wPlus`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `x`.

7 Click to expand the **Legends** section. Select the **Show legends** checkbox.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`Numerical, wall functions`
`Numerical, blended`
`Numerical, low Reynolds`

10 In the **Wall y+** toolbar, click **Plot**.

Finally, follow the instructions below to compare the skin friction and heat transfer coefficients obtained from numerical results with the ones computed from correlations, and reproduce the plot of Figure 3 and Figure 4.

Skin Friction Coefficient

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

2 In the **Settings** window for **1D Plot Group**, type `Skin Friction Coefficient` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol3)**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Skin friction coefficient`.

6 Locate the **Plot Settings** section.

7 Select the **y-axis label** checkbox. In the associated text field, type `\$C_f\$ (1)`.

8 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

9 In the **x minimum** text field, type `0`.

10 In the **x maximum** text field, type `5`.

11 In the **y minimum** text field, type `1e-3`.

12 In the **y maximum** text field, type `8e-3`.

Numerical

1 Right-click **Skin Friction Coefficient** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, type `Numerical` in the **Label** text field.

3 Select Boundary 2 only.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `Cf_film`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `x`.

7 Locate the **Legends** section. Select the **Show legends** checkbox.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`Numerical, wall functions`
`Numerical, blended`
`Numerical, low Reynolds`

White's correlation

1 In the **Model Builder** window, right-click **Skin Friction Coefficient** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol3)**.

4 From the **Parameter selection (mesh_coeff)** list, choose **Last**.

5 Select Boundary 2 only.

6 In the **Label** text field, type `White's correlation`.

7 Locate the **y-Axis Data** section. In the **Expression** text field, type `Cf_White`.

8 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

9 In the **Expression** text field, type `x`.

10 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

11 From the **Color** list, choose **From theme**.

12 Locate the **Legends** section. Select the **Show legends** checkbox.

13 Find the **Include** subsection. Clear the **Solution** checkbox.

14 Select the **Label** checkbox.

15 In the **Skin Friction Coefficient** toolbar, click **Plot**.

Heat Transfer Coefficient

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

2 In the **Settings** window for **1D Plot Group**, type `Heat Transfer Coefficient` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol3)**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Heat transfer coefficient`.

6 Locate the **Plot Settings** section.

7 Select the **y-axis label** checkbox. In the associated text field, type `\$h\$ (W/(m²·K))`.

8 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

9 In the **x minimum** text field, type `0`.

10 In the **x maximum** text field, type `5`.

11 In the **y minimum** text field, type `30`.

12 In the **y maximum** text field, type `150`.

Numerical

1 Right-click **Heat Transfer Coefficient** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, type `Numerical` in the **Label** text field.

3 Select Boundary 2 only.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `qw/(ht.Tu-Tb)`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `x`.

7 Locate the **Legends** section. Select the **Show legends** checkbox.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`Numerical, wall functions`
`Numerical, blended`
`Numerical, low Reynolds`

Lienhard's correlation

1 In the **Model Builder** window, right-click **Heat Transfer Coefficient** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol3)**.

4 From the **Parameter selection (mesh_coeff)** list, choose **Last**.

5 Select Boundary 2 only.

6 In the **Label** text field, type `Lienhard's correlation`.

7 Locate the **y-Axis Data** section. In the **Expression** text field, type `ht.kxx*Nu_x_turb_Lienhard/x`.

8 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

9 In the **Expression** text field, type `x`.

10 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

11 From the **Color** list, choose **From theme**.

12 Locate the **Legends** section. Select the **Show legends** checkbox.

13 Find the **Include** subsection. Clear the **Solution** checkbox.

14 Select the **Label** checkbox.

15 In the **Heat Transfer Coefficient** toolbar, click **Plot**.