Created in COMSOL Multiphysics 6.2

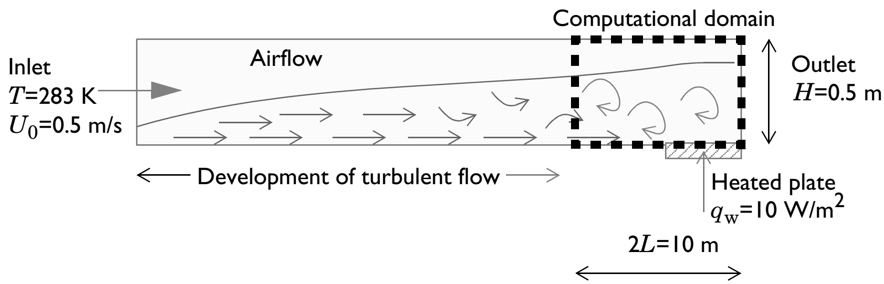
Nonisothermal Turbulent Flow over a Flat Plate

Introduction

This model of turbulent airflow over a flat plate validates the heat-transfer coefficient obtained from the simulation against Nusselt number based correlation functions. The simulation results are in good agreement with experimental measurements.

Model Definition

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



A plate of length L is heated with a constant heat flux q_w of 10 W/m^2 and placed in a turbulent airflow with average velocity U_0 = 0.5 m/s and temperature T_0 = 283 K. At the left boundary of the computational domain, the airflow turbulence is supposed to be fully develop. The airflow is heated over the plate.

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 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.

In addition, the **Fully developed** option of the **Inlet** boundary condition is used to set the turbulent inlet at the left boundary of the computational domain.

Nusselt Number Correlations

The following Nusselt number correlations are used to validate the numerical results.

In Ref. 1, p. 260, the Nusselt number Nu_x at the position x along the heated plate is defined as follows:

$$\mathrm{Nu}_x = 0.0296\mathrm{Re}_x^{4/5}\mathrm{Pr}^{1/3}$$

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:

$$\mathrm{Re}_x = \frac{\rho(T_{\mathrm{f},x})U_0x}{\mu(T_{\mathrm{f},x})}$$

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

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

where $\rho(T_{\mathrm{f},x})$ (SI unit: $\mathrm{kg/m^3}$) denotes the density, $\mu(T_{\mathrm{f},x})$ (SI unit: Pa·s) the viscosity, $C_p(T_{\mathrm{f},x})$ (SI unit: J/(kg·K)) the heat capacity, 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.

This correlation, initially developed for an isothermal wall, works satisfactorily when the heat flux is uniform, as mentioned in Ref. 1. It is valid for $\mathrm{Pr} > 0.5$.

In Ref. 2, p.327, a slightly different correlation is proposed for the Nusselt number:

$$\mathrm{Nu}_x = 0.032\mathrm{Re}_x^{4/5}\mathrm{Pr}^{0.43}$$

for flows such that $2\cdot10^5 < \mathrm{Re}_x < 5\cdot10^6$.

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

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

It is compared to the heat-transfer coefficient obtained from numerical simulation:

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

where $T_{\mathrm{b},x}$ 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

The velocity field and the temperature field over the plate are shown in Figure 1 in Figure 2 respectively.

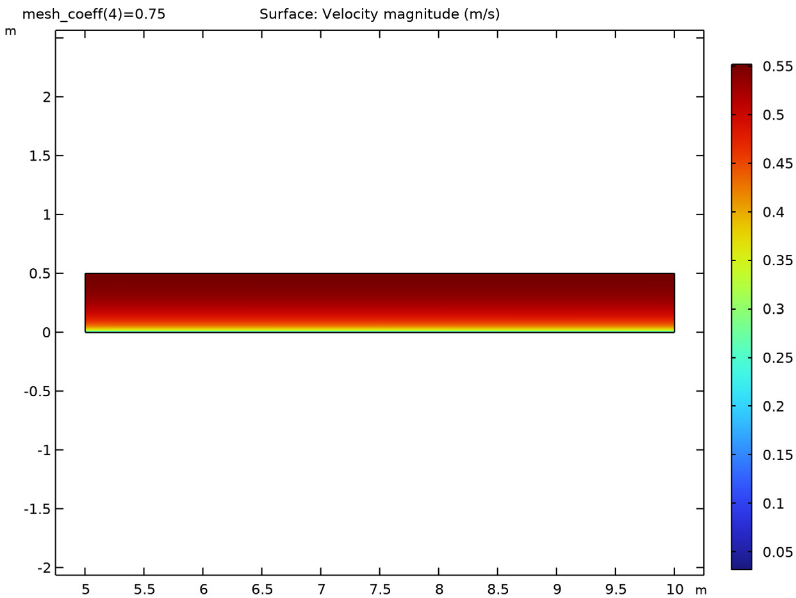


Figure 1: Velocity field over the plate (x > 5 m).

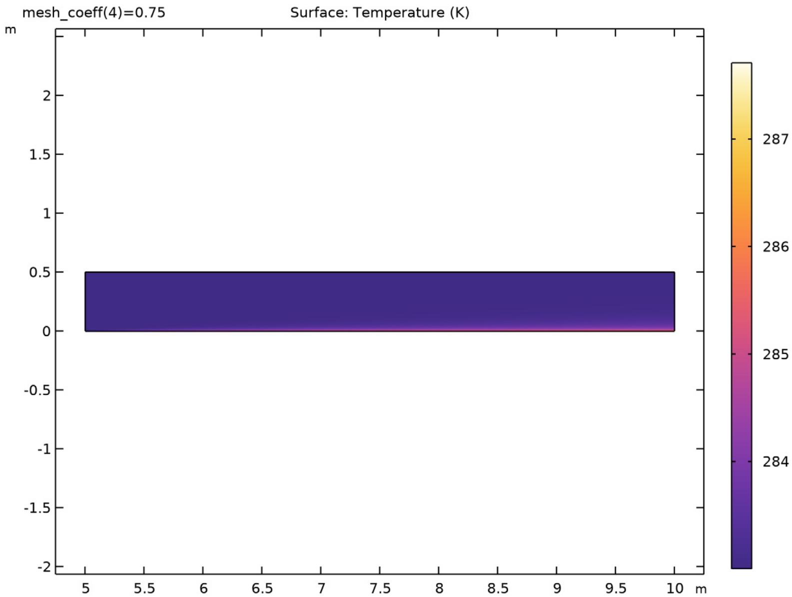


Figure 2: Temperature field over the plate (x > 5 m).

A numerical convergence study based on the mesh refinement is run using the `mesh_coeff` parameter. The comparison of the computed heat-transfer coefficient with the one obtained from the Nusselt number correlations shows a good approximation over the plate, for `mesh_coeff`> 0.1 (Figure 3). Further refinement of the mesh does not bring any significant improvement of the numerical solution.

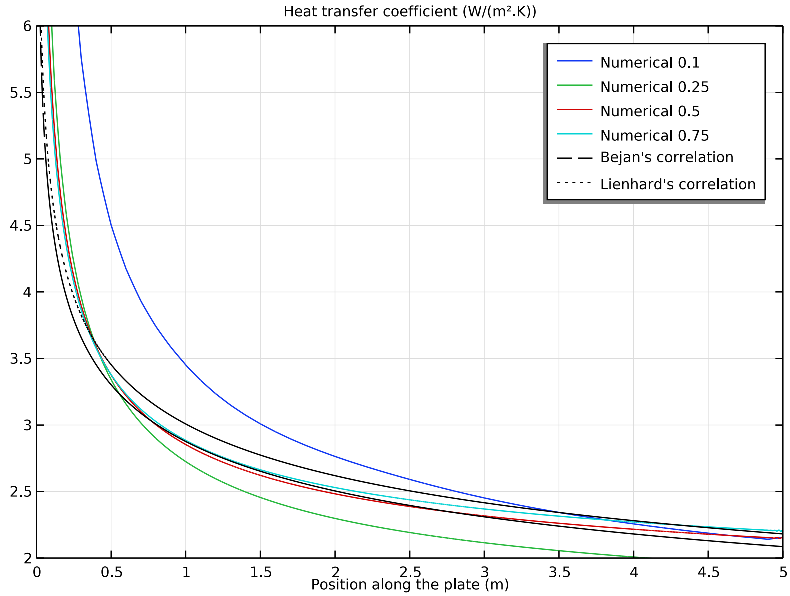


Figure 3: Comparison of the computed heat-transfer coefficient with the heat-transfer coefficient estimations based on Nusselt number correlations.

References

1. A. Bejan and others, *Heat Transfer Handbook*, John Wiley & Sons, 2003.

2. J.H. Lienhard IV and J.H. Lienhard V, *A Heat Transfer Textbook*, 4th edition, Phlogiston Press, 2017.

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	0.5[m/s]	0.5 m/s	Inlet velocity
qw	10[W/m^2]	10 W/m²	Wall heat flux
mesh_coeff	0.1	0.1	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*2.

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

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

Layer name	Thickness (m)
Layer 1	L

6 Clear the **Layers on bottom** check box.

7 Select the **Layers to the left** check box.

8 Click **Build All Objects**.

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 Click **Add to Component** in the window toolbar.

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

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
x_plate	x-L	m	Position along the plate
T_film	0.5*(T+T0)	K	Film temperature
rho_film	mat1.def.rho(ht.pA,T_film)	kg/m³	Film density

Name	Expression	Unit	Description
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
Pr_film	Cp_film*mu_film/k_film		Prandtl number based on film properties
Re_film	rho_film*U0*x_plate/mu_film		Reynolds number based on film properties
Nu_x_turb_Bejan	0.0296*Re_film^0.8*Pr_film^(1/3)		Nusselt number (Bejan, 5.131')
Nu_x_turb_Lienhard	0.032*Re_film^0.8*Pr_film^0.43		Nusselt number (Lienhard, 6.115)

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 **Boundary Condition** section.

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

5 Locate the **Fully Developed Flow** section. In the U_{av} text field, type `U0`.

Outlet 1

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

2 Select Boundary 7 only.

Symmetry 1

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

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

2 Select Boundaries 3 and 6 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 7 only.

Symmetry 1

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

2 Select Boundaries 3 and 6 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 5 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, under **Component 1 (comp1)>Mesh 1** right-click **Corner Refinement 1** and choose **Delete**.

2 Click **Yes** to confirm.

Free Triangular 1

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

2 Click **Yes** to confirm.

Boundary Layers 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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, 3, 5, and 6 only.

4 Locate the **Distribution** section. In the **Number of elements** text field, type `L*100*mesh_coeff`.

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

Distribution (vertical)

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

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, 4, and 7 only.

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

6 In the **Number of elements** text field, type `100*mesh_coeff`.

7 In the **Element ratio** text field, type `8`.

8 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)	0.1 0.25 0.5 0.75	

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

RESULTS

Velocity (spf)

The default plot groups show the **Velocity** and **Temperature** surface plots. Follow the instructions below to plot the distributions only over the plate, for a better visualization of the results, and to reproduce the plots shown in Figure 1 and Figure 2.

Study 1/Solution 1 (4) (sol1)

1 In the **Results** toolbar, click **More Datasets** and choose **Solution**.

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

3 From the **Solution** list, choose **Parametric Solutions 1 (sol3)**.

Selection

1 Right-click **Study 1/Parametric Solutions 1 (4) (sol3)** 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 Select Domain 2 only.

Velocity (spf)

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

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (4) (sol3)**.

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

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

Temperature (ht)

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

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (4) (sol3)**.

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

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

Heat Transfer Coefficient

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

Finally, follow the instructions below to compare the heat transfer coefficient obtained from numerical results with the one computed from a Nusselt correlation, and reproduce the plot of Figure 3.

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 (3) (sol3)**.

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

5 In the **Title** text area, type `Heat transfer coefficient (W/(m^2.K))`.

6 Locate the **Axis** section. Select the **Manual axis limits** check box.

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

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

9 In the **y minimum** text field, type `2`.

10 In the **y maximum** text field, type `6`.

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 5 only.

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

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

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

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

8 Find the **Include** subsection. Select the **Label** check box.

Bejan'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 (3) (sol3)**.

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

5 Select Boundary 5 only.

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

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

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

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

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** check box.
13. Find the **Include** subsection. Clear the **Solution** check box.
14. Select the **Label** check box.

Lienhard's correlation

1. 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 (3) (sol3)**.
4. From the **Parameter selection (mesh_coeff)** list, choose **Last**.
5. Select Boundary 5 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_plate`.
8. Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
9. In the **Expression** text field, type `x_plate`.
10. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.
11. From the **Color** list, choose **From theme**.
12. Locate the **Legends** section. Select the **Show legends** check box.
13. Find the **Include** subsection. Clear the **Solution** check box.
14. Select the **Label** check box.
15. In the **Heat Transfer Coefficient** toolbar, click **Plot**.