Created in COMSOL Multiphysics 6.2

Eppler Airfoil Transition

Introduction

This example simulates the flow around the Eppler 387 airfoil using the SST turbulence model both with and without the transition model. The results are compared with experimental values from Ref. 1.

Model Definition

The geometry of the Eppler 387 airfoil is taken from Ref. 2. The flow domain is composed of a semicircle with radius 100 m and a rectangle with width 100 m.

Figure 1 shows the flow domain and the applied far-field boundary conditions,

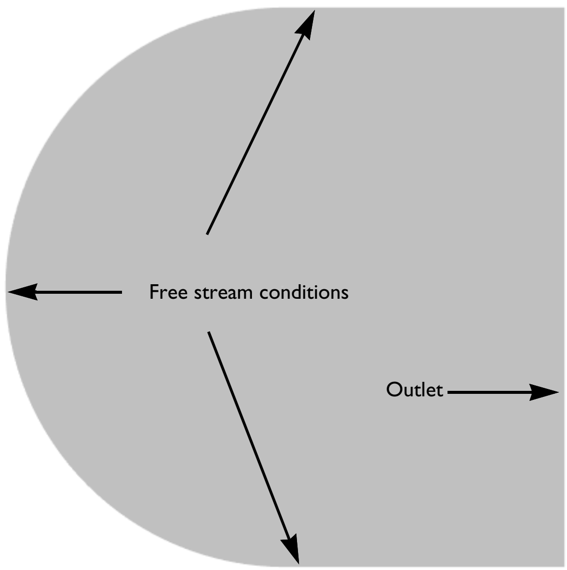


Figure 1: Flow domain and far-field boundary conditions.

Figure 2 shows a close-up of the airfoil profile. A no-slip condition with low-Re wall treatment is applied on the surface of the airfoil for the SST simulations.

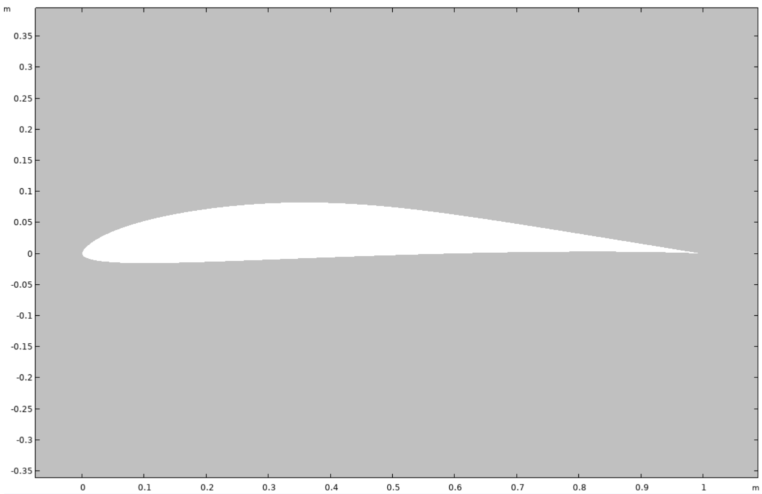


Figure 2: Close-up of the airfoil section.

A potential-flow solution was used to initialize the stationary SST simulation for the case without transition. Stationary solutions could not be obtained when the transition model was activated. Instead, transient simulations initialized with the stationary solutions without transition, were performed. An initial period of 4 s (roughly 4 by-pass times) was first simulated, and following that, data was collected during an additional period of 4 s. Time-averaging was then used to compare the results with the transition model to those without it.

Results and Discussion

The study performs a Parametric Sweep with the angle of attack taking the values

$$\alpha = 0°, 2°, 4°, 6°$$

Figure 3 shows the velocity magnitude around the Eppler 387 airfoil profile at the angle of attack $\alpha = 6°$.

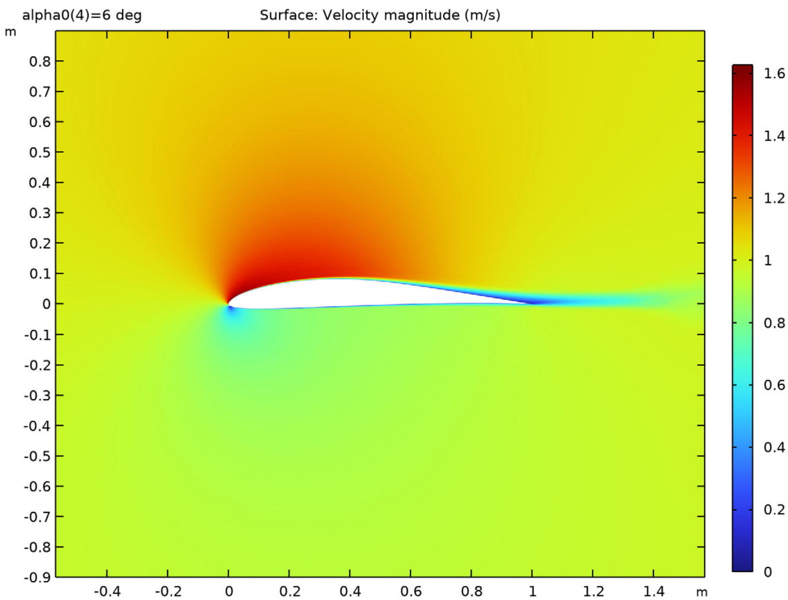


Figure 3: Flow field around the E387 airfoil at t = 8 s and $\alpha = 6°$.

Comparisons of pressure coefficients are presented at different angles of attack. The pressure coefficient is defined as

$$c_p(x) = \frac{p(x) - p_\infty}{\frac{1}{2}\rho_\infty U_\infty^2}$$

Figure 4 shows comparisons between the computed pressure coefficients, with and without the transition model, and the experimental results, Ref. 2, at different α.

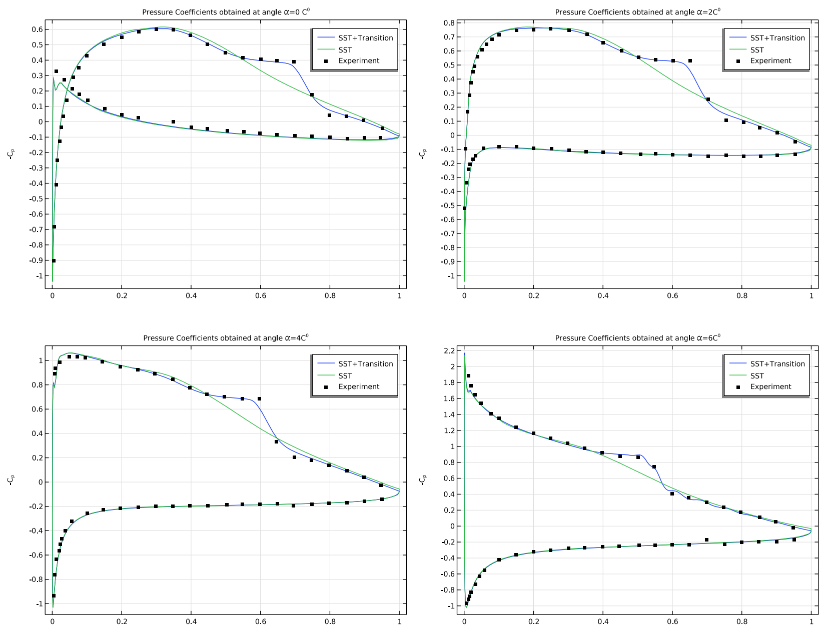


Figure 4: Comparison of pressure coefficients obtained with experimental data at different angles of attack α.

The SST turbulence model with transition modeling is able to capture the separation induced transition, and shows good agreement with the experimental results.

References

1. S. Karabay, "Implementation and assessment of K-Omega-Gamma transition model for turbulent flows," *NASA Langley Research Center Hampton, Virginia*, 1990.

2. G.M. Cole and T.J. Muller, "Experimental measurements of the laminar separation bubble on an Eppler 387 airfoil at low Reynolds numbers," *NASA Langley Research Center Hampton, Virginia*, 1990; https://ntrs.nasa.gov/citations/19900006064.

Application Library path: `CFD_Module/Verification_Examples/eppler_airfoil_transition`

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>Single-Phase Flow>Potential Flow>Incompressible Potential Flow (ipf)**.

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 In the table, enter the following settings:

Name	Expression	Value	Description
U0	1[m/s]	1 m/s	Free-stream velocity
rho0	1[kg/m^3]	1 kg/m³	Density
mu0	5e-6[Pa*s]	5E-6 Pa·s	Dynamic viscosity
k0	3/2*(0.18*U0)^2	0.0486 m²/s²	Free-stream turbulence kinetic energy
muT0	2*mu0	1E-5 Pa·s	Free-stream turbulence dynamic viscosity
om0	rho0*k0/mu0	9720 1/s	Free-stream turbulence specific energy dissipation
alpha0	0[deg]	0 rad	Angle of attack
chord	1[m]	1 m	Chord length

GEOMETRY 1

Interpolation Curve 1 (ic1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Interpolation Curve**.

2 In the **Settings** window for **Interpolation Curve**, locate the **Object Type** section.

3 From the **Type** list, choose **Solid**.

4 Locate the **Interpolation Points** section. Click **Load from File**.

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

6 Click **Build Selected**.

Offset 1 (off1)

1 In the **Geometry** toolbar, click **Offset**.

2 Select the object **ic1** only.

3 In the **Settings** window for **Offset**, locate the **Options** section.

4 In the **Distance** text field, type `0.25`.

5 Click **Build Selected**.

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.

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

3 In the **Radius** text field, type `100`.

4 In the **Sector angle** text field, type `180`.

5 Locate the **Position** section. In the **x** text field, type `1`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `90`.

7 Click **Build Selected**.

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 `100`.

4 In the **Height** text field, type `200`.

5 Locate the **Position** section. In the **x** text field, type `1`.

6 In the **y** text field, type `-100`.

7 Click **Build Selected**.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **x** text field, type `-0.25`.

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **x** text field, type `0.005`.

7 Click **Build Selected**.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the objects **c1**, **ls1**, **off1**, and **r1** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **ic1** only.

6 Click **Build All Objects**.

Ignore Vertices 1 (igv1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Vertices**.

2 On the object **fin**, select Points 3, 5, 6, and 12 only.

3 Right-click **Ignore Vertices 1 (igv1)** and choose **Build All Objects**.

DEFINITIONS

Integration 1 (intop1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Nonlocal Couplings>Integration**.

3 In the **Settings** window for **Integration**, locate the **Source Selection** section.

4 From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundaries 13 and 14 only.

Variables 1

1 Right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
C_lift_t	2*intop1(-spf2.T_stressy)/(rho0*U0^2*chord)		Lift coefficient, transient
C_lift	2*intop1(-spf.T_stressy)/(rho0*U0^2*chord)		Lift coefficient

INCOMPRESSIBLE POTENTIAL FLOW (IPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Incompressible Potential Flow (ipf)**.

2 In the **Settings** window for **Incompressible Potential Flow**, locate the **Pressure** section.

3 In the U_{scale} text field, type U0.

Fluid Properties 1

1 In the **Model Builder** window, expand the **Incompressible Potential Flow (ipf)** node, then click **Fluid Properties 1**.

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

3 From the ρ list, choose **User defined**. In the associated text field, type rho0.

Velocity 1

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

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

3 Select Boundaries 3, 7, 9, and 10 only.

4 In the **Settings** window for **Velocity**, locate the **Velocity** section.

5 In the U_{in} text field, type U0*(nxmesh*cos(alpha0)+nymesh*sin(alpha0)).

Open Boundary 1

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

2 Select Boundary 8 only.

MESH 1

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `5`.

5 In the **Maximum element growth rate** text field, type `1.2`.

Free Quad 1

1 In the **Mesh** toolbar, click **Free Quad**.

2 Right-click **Free Quad 1** and choose **Move Up**.

3 In the **Settings** window for **Free Quad**, locate the **Domain Selection** section.

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

5 Select Domains 1, 2, and 5 only.

6 Click to expand the **Control Entities** section. In the **Number of iterations** text field, type `10`.

7 In the **Maximum element depth to process** text field, type `8`.

Size 1

1 Right-click **Free Quad 1** and choose **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 **Extremely fine**.

5 Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `0.005`.

8 Select the **Minimum element size** check box. In the associated text field, type `0.001`.

Boundary Layers 1

1 In the **Mesh** toolbar, click **Boundary Layers**.

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

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

4 Select Domains 1–3 and 5 only.

Boundary Layer Properties

1 In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties**.

2 Select Boundaries 13 and 14 only.

3 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

4 In the **Number of layers** text field, type `60`.

5 In the **Stretching factor** text field, type `1.075`.

6 From the **Thickness specification** list, choose **First layer**.

7 In the **Thickness** text field, type `1e-5`.

8 Click **Build All**.

STUDY 1

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

ADD PHYSICS

1 In the **Home** toolbar, click **Windows** and choose **Add Physics**.

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

3 In the tree, select **Fluid Flow>Single-Phase Flow>Turbulent Flow>Turbulent Flow, SST (spf)**.

4 Click **Add to Component 1** in the window toolbar.

TURBULENT FLOW, SST (SPF)

1 In the **Settings** window for **Turbulent Flow, SST**, locate the **Turbulence** section.

2 From the **Wall treatment** list, choose **Low Re**.

Fluid Properties 1

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

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

3 From the ρ list, choose **User defined**. In the associated text field, type `rho0`.

4 From the μ list, choose **User defined**. In the associated text field, type `mu0`.

5 Locate the **Distance Equation** section. From the l_{ref} list, choose **Manual**.

6 In the text field, type `0.2`.

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 Specify the **u** vector as

`ipf.u`	x
`ipf.v`	y

4 In the *p* text field, type `ipf.p`.

5 In the *k* text field, type `k0`.

6 In the *om* text field, type `om0`.

Inlet 1

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

2 Select Boundaries 3, 7, 9, and 10 only.

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

4 Click the **Velocity field** button.

5 Specify the $\mathbf{u}_0$ vector as

`U0*cos(alpha0)`	x
`U0*sin(alpha0)`	y

6 Locate the **Turbulence Conditions** section. Click the **Specify turbulence variables** button.

7 In the k_0 text field, type `k0`.

8 In the ω_0 text field, type `om0`.

Open Boundary 1

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

2 Select Boundary 8 only.

3 In the **Settings** window for **Open Boundary**, locate the **Turbulence Conditions** section.

4 Click the **Specify turbulence variables** button.

5 In the k_0 text field, type `k0`.

6 In the ω_0 text field, type `om0`.

ADD STUDY

1 In the **Home** toolbar, click **Windows** and choose **Add Study**.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Turbulent Flow, SST>Stationary with Initialization**.

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

STUDY 2

Step 2: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 2: Stationary**.

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

3 In the table, clear the **Solve for** check box for **Incompressible Potential Flow (ipf)**.

4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
alpha0 (Angle of attack)	`0,2,4,6`	`deg`

Step 1: Wall Distance Initialization

1 In the **Model Builder** window, click **Step 1: Wall Distance Initialization**.

2 In the **Settings** window for **Wall Distance Initialization**, click to expand the **Values of Dependent Variables** section.

3 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

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

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 2** node, then click **Segregated 1**.

4 In the **Settings** window for **Segregated**, locate the **General** section.

5 In the **Maximum number of iterations** text field, type `400`.

6 In the **PID controller - derivative** text field, type `0.03`.

7 In the **Target error estimate** text field, type `0.08`.

8 Clear the **Anderson acceleration** check box.

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

ADD PHYSICS

1 In the **Home** toolbar, click **Windows** and choose **Add Physics**.
2 Go to the **Add Physics** window.
3 In the tree, select **Fluid Flow>Single-Phase Flow>Turbulent Flow>Turbulent Flow, SST (spf)**.
4 Click **Add to Component 1** in the window toolbar.

TURBULENT FLOW, SST 2 (SPF2)

1 In the **Settings** window for **Turbulent Flow, SST**, locate the **Turbulence** section.
2 Select the **Include transition modeling** check box.

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Turbulent Flow, SST 2 (spf2)** click **Fluid Properties 1**.
2 In the **Settings** window for **Fluid Properties**, locate the **Fluid Properties** section.
3 From the ρ list, choose **User defined**. In the associated text field, type `rho0`.
4 From the μ list, choose **User defined**. In the associated text field, type `mu0`.
5 Locate the **Distance Equation** section. From the l_{ref} list, choose **Manual**.
6 In the text field, type `0.2`.

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 Specify the **u** vector as

u	x
v	y

4 In the p text field, type `p`.
5 In the k text field, type `k`.
6 In the om text field, type `spf.om_global`.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2 Select Boundaries 3, 7, 9, and 10 only.
3 In the **Settings** window for **Inlet**, locate the **Velocity** section.
4 Click the **Velocity field** button.

5 Specify the $\mathbf{u}_0$ vector as

`U0*cos(alpha0)`	x
`U0*sin(alpha0)`	y

6 Locate the **Turbulence Conditions** section. Click the **Specify turbulence variables** button.

7 In the k_0 text field, type `k0`.

8 In the ω_0 text field, type `om0`.

Open Boundary 1

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

2 Select Boundary 8 only.

3 In the **Settings** window for **Open Boundary**, locate the **Turbulence Conditions** section.

4 Click the **Specify turbulence variables** button.

5 In the k_0 text field, type `k0`.

6 In the ω_0 text field, type `om0`.

7 In the γ_0 text field, type `1`.

ADD STUDY

1 In the **Home** toolbar, click **Windows** and choose **Add Study**.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Time Dependent with Initialization**.

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

STUDY 3

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
alpha0 (Angle of attack)	`0,2,4,6`	deg

Step 1: Wall Distance Initialization

1 In the **Model Builder** window, click **Step 1: Wall Distance Initialization**.

2 In the **Settings** window for **Wall Distance Initialization**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Turbulent Flow, SST (spf)**.

4 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Study** list, choose **Study 2, Stationary**.

6 From the **Solution** list, choose **Solution 2 (sol2)**.

Step 2: Time Dependent

1 In the **Model Builder** window, click **Step 2: Time Dependent**.

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

3 In the **Output times** text field, type `range(0,1,4), range(4,0.1,8)`.

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Turbulent Flow, SST (spf)**.

Solution 4 (sol4)

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

2 In the **Model Builder** window, expand the **Solution 4 (sol4)** 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 `0.0025`.

6 From the **Minimum BDF order** list, choose **2**.

7 Select the **Rescale after initialization** check box.

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

DEFINITIONS

View 1

In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

Axis

1 In the **Model Builder** window, expand the **View 1** node, then click **Axis**.

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

3 In the **x minimum** text field, type `-0.4`.

4 In the **x maximum** text field, type `1.4`.

5 In the **y minimum** text field, type `-0.4`.

6 In the **y maximum** text field, type `0.4`.

7 Click **Update**.

View 1

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

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

3 Select the **Lock axis** check box.

RESULTS

Time Average 1

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets** and choose **More Datasets>Time Average**.

3 In the **Settings** window for **Time Average**, locate the **Data** section.

4 From the **Dataset** list, choose **Exterior Walls 1**.

5 From the **Parameter selection (alpha0)** list, choose **First**.

6 From the **Time selection** list, choose **From list**.

7 In the **Times (s)** list, choose **4 (2)**, **4.1**, **4.2**, **4.3**, **4.4**, **4.5**, **4.6**, **4.7**, **4.8**, **4.9**, **5**, **5.1**, **5.2**, **5.3**, **5.4**, **5.5**, **5.6**, **5.7**, **5.8**, **5.9**, **6**, **6.1**, **6.2**, **6.3**, **6.4**, **6.5**, **6.6**, **6.7**, **6.8**, **6.9**, **7**, **7.1**, **7.2**, **7.3**, **7.4**, **7.5**, **7.6**, **7.7**, **7.8**, **7.9**, and **8**.

Time Average 2

1 In the **Results** toolbar, click **More Datasets** and choose **Evaluation>Time Average**.

2 In the **Settings** window for **Time Average**, locate the **Data** section.

3 From the **Dataset** list, choose **Exterior Walls 1**.

4 From the **Parameter selection (alpha0)** list, choose **From list**.

5 In the **Parameter values (alpha0 (deg))** list, select **2**.

6 From the **Time selection** list, choose **From list**.

7 In the **Times (s)** list, choose **4 (2)**, **4.1**, **4.2**, **4.3**, **4.4**, **4.5**, **4.6**, **4.7**, **4.8**, **4.9**, **5**, **5.1**, **5.2**, **5.3**, **5.4**, **5.5**, **5.6**, **5.7**, **5.8**, **5.9**, **6**, **6.1**, **6.2**, **6.3**, **6.4**, **6.5**, **6.6**, **6.7**, **6.8**, **6.9**, **7**, **7.1**, **7.2**, **7.3**, **7.4**, **7.5**, **7.6**, **7.7**, **7.8**, **7.9**, and **8**.

Time Average 3

1 In the **Results** toolbar, click **More Datasets** and choose **Evaluation>Time Average**.

2 In the **Settings** window for **Time Average**, locate the **Data** section.

3 From the **Dataset** list, choose **Exterior Walls 1**.

4 From the **Parameter selection (alpha0)** list, choose **From list**.

5 In the **Parameter values (alpha0 (deg))** list, select **4**.

6 From the **Time selection** list, choose **From list**.

7 In the **Times (s)** list, choose **4 (2)**, **4.1**, **4.2**, **4.3**, **4.4**, **4.5**, **4.6**, **4.7**, **4.8**, **4.9**, **5**, **5.1**, **5.2**, **5.3**, **5.4**, **5.5**, **5.6**, **5.7**, **5.8**, **5.9**, **6**, **6.1**, **6.2**, **6.3**, **6.4**, **6.5**, **6.6**, **6.7**, **6.8**, **6.9**, **7**, **7.1**, **7.2**, **7.3**, **7.4**, **7.5**, **7.6**, **7.7**, **7.8**, **7.9**, and **8**.

Time Average 4

1 In the **Results** toolbar, click **More Datasets** and choose **Evaluation>Time Average**.

2 In the **Settings** window for **Time Average**, locate the **Data** section.

3 From the **Dataset** list, choose **Exterior Walls 1**.

4 From the **Parameter selection (alpha0)** list, choose **From list**.

5 In the **Parameter values (alpha0 (deg))** list, select **6**.

6 From the **Time selection** list, choose **From list**.

7 In the **Times (s)** list, choose **4 (2)**, **4.1**, **4.2**, **4.3**, **4.4**, **4.5**, **4.6**, **4.7**, **4.8**, **4.9**, **5**, **5.1**, **5.2**, **5.3**, **5.4**, **5.5**, **5.6**, **5.7**, **5.8**, **5.9**, **6**, **6.1**, **6.2**, **6.3**, **6.4**, **6.5**, **6.6**, **6.7**, **6.8**, **6.9**, **7**, **7.1**, **7.2**, **7.3**, **7.4**, **7.5**, **7.6**, **7.7**, **7.8**, **7.9**, and **8**.

Lift coefficient stationary Aoa=0

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

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

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

4 From the **Parameter selection (alpha0)** list, choose **First**.

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

Expression	Unit	Description
`C_lift`	1	`Lift coefficient`

6 In the **Label** text field, type `Lift coefficient stationary Aoa=0`.

7 Right-click **Lift coefficient stationary Aoa=0** and choose **Duplicate**.

Lift coefficient stationary Aoa=2

1. In the **Model Builder** window, under **Results>Derived Values** click **Lift coefficient stationary Aoa=0.1**.
2. In the **Settings** window for **Global Evaluation**, type `Lift coefficient stationary Aoa=2` in the **Label** text field.
3. Locate the **Data** section. From the **Parameter selection (alpha0)** list, choose **From list**.
4. In the **Parameter values (alpha0 (deg))** list, select **2**.
5. Right-click **Lift coefficient stationary Aoa=2** and choose **Duplicate**.

Lift coefficient stationary Aoa=4

1. In the **Model Builder** window, click **Lift coefficient stationary Aoa=2.1**.
2. In the **Settings** window for **Global Evaluation**, locate the **Data** section.
3. In the **Parameter values (alpha0 (deg))** list, select **4**.
4. In the **Label** text field, type `Lift coefficient stationary Aoa=4`.
5. Right-click **Lift coefficient stationary Aoa=4** and choose **Duplicate**.

Lift coefficient stationary Aoa=6

1. In the **Model Builder** window, click **Lift coefficient stationary Aoa=4.1**.
2. In the **Settings** window for **Global Evaluation**, locate the **Data** section.
3. In the **Parameter values (alpha0 (deg))** list, select **6**.
4. In the **Label** text field, type `Lift coefficient stationary Aoa=6`.
5. Right-click **Lift coefficient stationary Aoa=6** and choose **Duplicate**.

Lift coefficient transient Aoa=0

1. In the **Model Builder** window, click **Lift coefficient stationary Aoa=6.1**.
2. In the **Settings** window for **Global Evaluation**, locate the **Data** section.
3. From the **Dataset** list, choose **Time Average 1**.
4. In the **Label** text field, type `Lift coefficient transient Aoa=0`.
5. Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
C_lift_t	1	Lift coefficient, transient

6. Right-click **Lift coefficient transient Aoa=0** and choose **Duplicate**.

Lift coefficient transient Aoa=2

1 In the **Model Builder** window, under **Results>Derived Values** click **Lift coefficient transient Aoa=0.1**.
2 In the **Settings** window for **Global Evaluation**, type `Lift coefficient transient Aoa=2` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Time Average 2**.
4 Right-click **Lift coefficient transient Aoa=2** and choose **Duplicate**.

Lift coefficient transient Aoa=4

1 In the **Model Builder** window, under **Results>Derived Values** click **Lift coefficient transient Aoa=2.1**.
2 In the **Settings** window for **Global Evaluation**, type `Lift coefficient transient Aoa=4` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Time Average 3**.
4 Right-click **Lift coefficient transient Aoa=4** and choose **Duplicate**.

Lift coefficient transient Aoa=6

1 In the **Model Builder** window, under **Results>Derived Values** click **Lift coefficient transient Aoa=4.1**.
2 In the **Settings** window for **Global Evaluation**, type `Lift coefficient transient Aoa=6` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Time Average 4**.

Exp. Aoa=0

1 In the **Results** toolbar, click **Table**.
2 In the **Settings** window for **Table**, type `Exp. Aoa=0` in the **Label** text field.
3 Locate the **Data** section. Click **Import**.
4 Browse to the model's Application Libraries folder and double-click the file `eppler_airfoil_transition_AOa_0_res.txt`.

Exp. Aoa=2

1 In the **Results** toolbar, click **Table**.
2 In the **Settings** window for **Table**, type `Exp. Aoa=2` in the **Label** text field.
3 Locate the **Data** section. Click **Import**.
4 Browse to the model's Application Libraries folder and double-click the file `eppler_airfoil_transition_AOa_2_res.txt`.

Exp. Aoa=4

1 In the **Results** toolbar, click **Table**.
2 In the **Settings** window for **Table**, type `Exp. Aoa=4` in the **Label** text field.
3 Locate the **Data** section. Click **Import**.
4 Browse to the model's Application Libraries folder and double-click the file `eppler_airfoil_transition_AOa_4_res.txt`.

Exp. Aoa=6

1 In the **Results** toolbar, click **Table**.
2 In the **Settings** window for **Table**, type `Exp. Aoa=6` in the **Label** text field.
3 Locate the **Data** section. Click **Import**.
4 Browse to the model's Application Libraries folder and double-click the file `eppler_airfoil_transition_AOa_6_res.txt`.

Wall Resolution (spf)

1 In the **Model Builder** window, under **Results** click **Wall Resolution (spf)**.
2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3 From the **Parameter value (alpha0 (deg))** list, choose **0**.

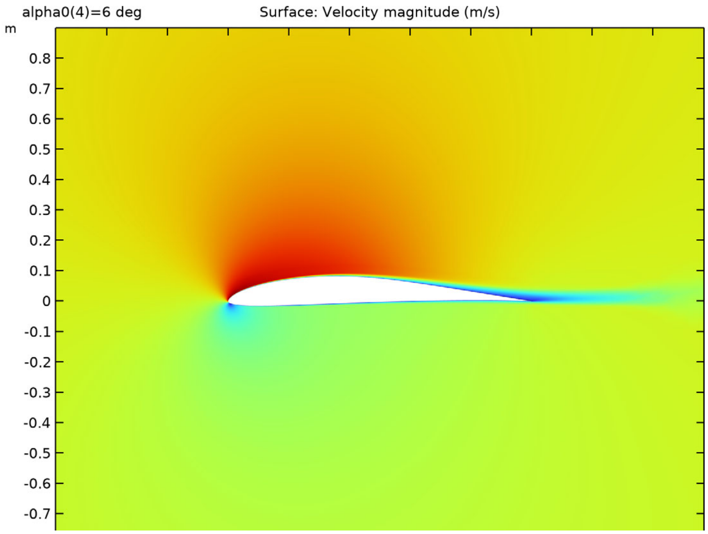
Wall Resolution

1 In the **Model Builder** window, expand the **Wall Resolution (spf)** node, then click **Wall Resolution**.
2 In the **Settings** window for **Line**, locate the **Coloring and Style** section.
3 In the **Tube radius expression** text field, type `0.01`.
4 Select the **Radius scale factor** check box. In the associated text field, type `1`.
5 In the **Wall Resolution (spf)** toolbar, click **Plot**.

Velocity (spf2)

1 In the **Model Builder** window, under **Results** click **Velocity (spf2)**.
2 In the **Settings** window for **2D Plot Group**, locate the **Plot Settings** section.
3 Clear the **Plot dataset edges** check box.
4 In the **Graphics** window toolbar, click ▾ next to **Go to Default View**, then choose **Go to View 1**.

5 In the **Velocity (spf2)** toolbar, click **Plot**.



Wall Resolution

1 In the **Model Builder** window, expand the **Wall Resolution (spf2)** node, then click **Wall Resolution**.

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

3 In the **Tube radius expression** text field, type `0.01`.

4 Select the **Radius scale factor** check box. In the associated text field, type `1`.

5 In the **Wall Resolution (spf2)** toolbar, click **Plot**.

Aoa=0

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

2 In the **Settings** window for **1D Plot Group**, type `Aoa=0` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** check box. In the associated text field, type `-C<sub>p</sub>`.

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

6 In the **Title** text area, type `Pressure Coefficients obtained at angle \alpha =0 C<sup>0</sup>`.

Line Graph 1

Right-click **Aoa=0** and choose **Line Graph**.

SST+Transition

1 In the **Model Builder** window, expand the **Results>Wall Resolution (spf2)>Wall Resolution** node, then click **Results>Aoa=0>Line Graph 1**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Time Average 1**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `-p2/(rho0*U0^2/2)`.

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** check box.

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

9 In the table, enter the following settings:

Legends
SST+Transition

SST

1 In the **Model Builder** window, right-click **Aoa=0** and choose **Line Graph**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Exterior Walls**.

4 From the **Parameter selection (alpha0)** list, choose **First**.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `-p/(rho0*U0^2/2)`.

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

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

8 Locate the **Legends** section. Select the **Show legends** check box.

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

10 In the table, enter the following settings:

Legends
SST

Experiment

1 Right-click **Aoa=0** and choose **Table Graph**.

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

3 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

4 From the **Color** list, choose **Black**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

6 Click to expand 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
Experiment

9 In the **Aoa=0** toolbar, click **Plot**.

Aoa=0

Right-click **Aoa=0** and choose **Duplicate**.

Aoa=2

1 In the **Model Builder** window, under **Results** click **Aoa=0.1**.

2 In the **Settings** window for **1D Plot Group**, type Aoa=2 in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type Pressure Coefficients obtained at angle \alpha =2C⁰.

SST+Transition

1 In the **Model Builder** window, expand the **Aoa=2** node, then click **SST+Transition**.

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

3 From the **Dataset** list, choose **Time Average 2**.

SST

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

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

3 From the **Parameter selection (alpha0)** list, choose **From list**.

4 In the **Parameter values (alpha0 (deg))** list, select **2**.

Experiment

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

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

3 From the **Table** list, choose **Exp. Aoa=2**.

4 In the **Aoa=2** toolbar, click **Plot**.

Aoa=2

In the **Model Builder** window, right-click **Aoa=2** and choose **Duplicate**.

Aoa=4

1 In the **Model Builder** window, under **Results** click **Aoa=2.1**.
2 In the **Settings** window for **1D Plot Group**, type Aoa=4 in the **Label** text field.
3 Locate the **Title** section. In the **Title** text area, type Pressure Coefficients obtained at angle \alpha =4C⁰.

SST+Transition

1 In the **Model Builder** window, expand the **Aoa=4** node, then click **SST+Transition**.
2 In the **Settings** window for **Line Graph**, locate the **Data** section.
3 From the **Dataset** list, choose **Time Average 3**.

SST

1 In the **Model Builder** window, click **SST**.
2 In the **Settings** window for **Line Graph**, locate the **Data** section.
3 In the **Parameter values (alpha0 (deg))** list, select **4**.

Experiment

1 In the **Model Builder** window, click **Experiment**.
2 In the **Settings** window for **Table Graph**, locate the **Data** section.
3 From the **Table** list, choose **Exp. Aoa=4**.
4 In the **Aoa=4** toolbar, click **Plot**.

Aoa=4

In the **Model Builder** window, right-click **Aoa=4** and choose **Duplicate**.

Aoa=6

1 In the **Model Builder** window, under **Results** click **Aoa=4.1**.
2 In the **Settings** window for **1D Plot Group**, type Aoa=6 in the **Label** text field.
3 Locate the **Title** section. In the **Title** text area, type Pressure Coefficients obtained at angle \alpha =6C⁰.

SST+Transition

1 In the **Model Builder** window, expand the **Aoa=6** node, then click **SST+Transition**.
2 In the **Settings** window for **Line Graph**, locate the **Data** section.
3 From the **Dataset** list, choose **Time Average 4**.

SST

1 In the **Model Builder** window, click **SST**.
2 In the **Settings** window for **Line Graph**, locate the **Data** section.
3 In the **Parameter values (alpha0 (deg))** list, select **6**.

Experiment

1 In the **Model Builder** window, click **Experiment**.
2 In the **Settings** window for **Table Graph**, locate the **Data** section.
3 From the **Table** list, choose **Exp. Aoa=6**.
4 In the **Aoa=6** toolbar, click **Plot**.