



Model created in COMSOL Multiphysics 6.4

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,

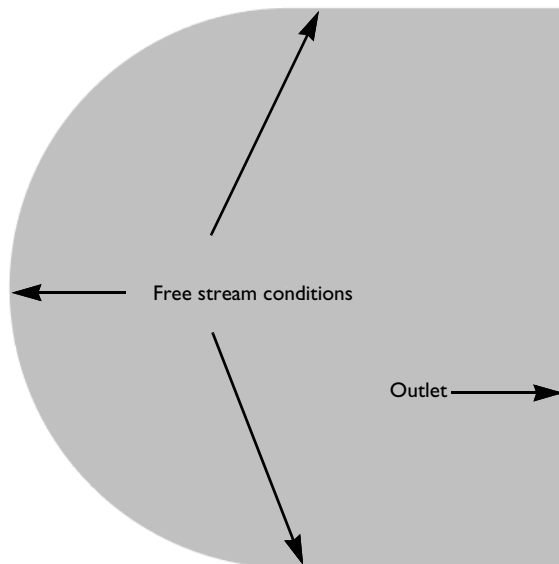


Figure 1: Flow domain and far-field boundary conditions.

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.

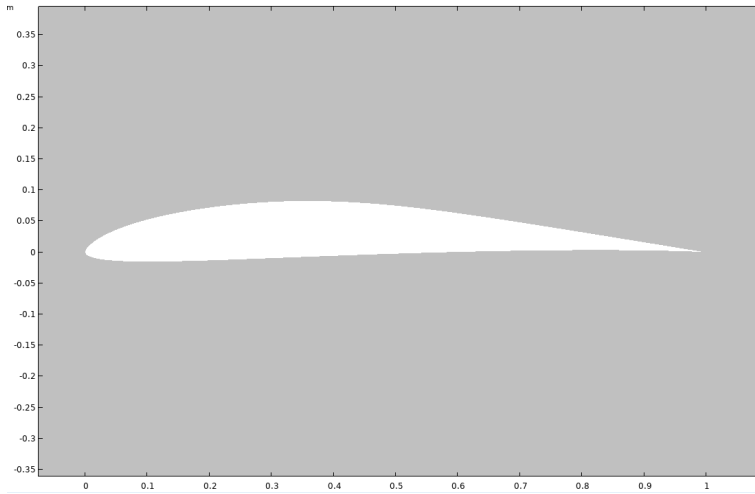


Figure 2: Close-up of the airfoil section.

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

$$\alpha = 0^\circ, 2^\circ, 4^\circ, 6^\circ$$

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

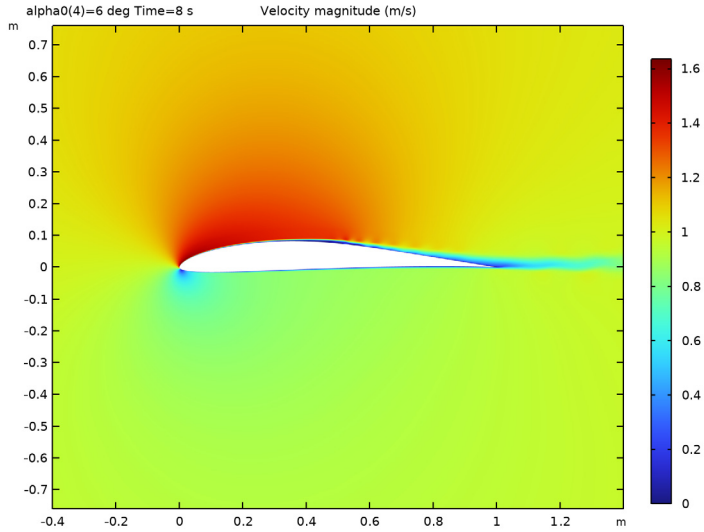


Figure 3: Flow field around the E387 airfoil at $t = 8$ s and $\alpha = 6^\circ$.

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 .

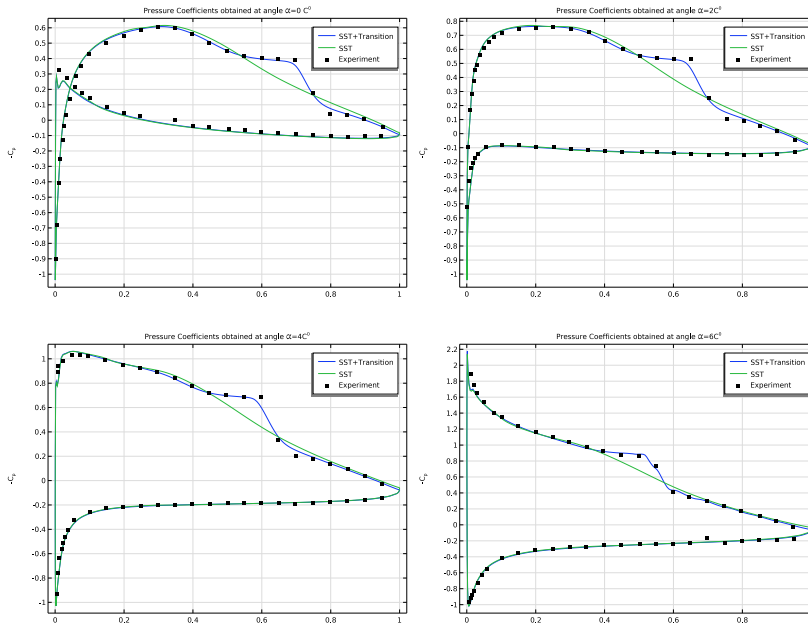


Figure 4: Comparison of pressure coefficients obtained with experimental data at different angles of attack α .

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; 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 I

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 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/m ³	Density
mu0	5e-6[Pa*s]	5E-6 Pa*s	Dynamic viscosity
k0	3/2*(0.18*U0)^2	0.0486 m ² /s ²	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 I



Interpolation Curve I (icI)

- 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 I (offI)

- 1 In the **Geometry** toolbar, click  **Offset**.
- 2 Select the object **icI** 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 I (cI)

- 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 I (rI)

- 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 I (lsI)

- 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 I (difI)

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.
- 2 Select the objects **cl**, **lsI**, **offI**, and **rl** 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 **icI** only.
- 6 Click  **Build All Objects**.

Ignore Vertices I (igvI)

- 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 I (igvI)** and choose **Build All Objects**.

DEFINITIONS

Integration I (intopI)

- 1 In the **Model Builder** window, expand the **Component I (compI)** > **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 I

- 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	$2 \cdot \text{intop1}(-\text{spf2.T_stressy}) / (\rho_0 \cdot U_0^2 \cdot \text{chord})$		Lift coefficient, transient
C_lift	$2 \cdot \text{intop1}(-\text{spf.T_stressy}) / (\rho_0 \cdot U_0^2 \cdot \text{chord})$		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 U_0 .

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 ρ_0 .

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 $U_0 \cdot (\text{nxmesh} \cdot \cos(\alpha_0) + \text{nymesh} \cdot \sin(\alpha_0))$.

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.

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** checkbox. In the associated text field, type 0.005.
- 8 Select the **Minimum element size** checkbox. 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 I

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

ADD PHYSICS

- 1 In the **Home** toolbar, click  **Add Physics** to open the **Add Physics** window.
- 2 Go to the **Add Physics** window.
- 3 In the tree, select **Fluid Flow** > **Single-Phase Flow** > **Turbulent Flow** > **Turbulent Flow, SST (spf)**.
- 4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Study I**.
- 5 Click the **Add to Component 1** button in the window toolbar.
- 6 In the **Home** toolbar, click  **Add Physics** to close the **Add Physics** window.

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 ρ_{00} .
- 4 From the μ list, choose **User defined**. In the associated text field, type μ_0 .
- 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 \mathbf{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 ω text field, type $\omega 0$.

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

$U0*\cos(\alpha 0)$	x
$U0*\sin(\alpha 0)$	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 $\omega 0$.

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 $\omega 0$.

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 **Preset Studies for Selected Physics Interfaces > Turbulent Flow, SST > Stationary with Initialization**.

- 4 Click the **Add Study** button in the window toolbar.
- 5 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Physics	Solve
off	√

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

STUDY 2

Step 2: Stationary


- 1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 2 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Incompressible Potential Flow (ipf)**.
- 3 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.
- 4 Click **+ Add**.
- 5 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** checkbox.
- 9 In the **Model Builder** window, expand the **Study 2 > Solver Configurations > Solution 2 (sol2) > Stationary Solver 2 > Segregated 1** node, then click **Turbulence Variables**.
- 10 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.
- 11 Clear the **Use adaptive tolerance in the linear solver** checkbox.
- 12 In the **Study** toolbar, click  **Compute**.

ADD PHYSICS

- 1 In the **Home** toolbar, click  **Add Physics** to open the **Add Physics** window.
- 2 Go to the **Add Physics** window.
- 3 In the tree, select **Fluid Flow > Single-Phase Flow > Turbulent Flow > Turbulent Flow, SST (spf)**.
- 4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkboxes for **Study 1** and **Study 2**.
- 5 Click the **Add to Component 1** button in the window toolbar.
- 6 In the **Home** toolbar, click  **Add Physics** to close the **Add Physics** window.

TURBULENT FLOW, SST 2 (SPF2)

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

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 \mathbf{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 ω text field, type $\text{spf.}\omega_{\text{global}}$.

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

$U_0 \cdot \cos(\alpha_0)$	x
$U_0 \cdot \sin(\alpha_0)$	y

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

7 In the k_0 text field, type k_0 .

8 In the ω_0 text field, type ω_0 .

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 k_0 .

6 In the ω_0 text field, type ω_0 .

7 In the γ_0 text field, type 1.


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

Preset Studies for Some Physics Interfaces > Time Dependent with Initialization.

- 4 Click the **Add Study** button in the window toolbar.
- 5 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkboxes for **Incompressible Potential Flow (ipf)** and **Turbulent Flow, SST (spf)**.
- 6 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.

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 **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox 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 **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox 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** checkbox.
- 8 In the **Study** toolbar, click  **Compute**.

RESULTS


Exterior Walls

- 1 In the **Model Builder** window, expand the **Results > Datasets** node.
- 2 Right-click **Results > Datasets** and choose **Edge 2D**.
- 3 In the **Settings** window for **Edge 2D**, locate the **Data** section.
- 4 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
- 5 In the **Label** text field, type Exterior Walls.
- 6 Select Boundaries 13 and 14 only.


Exterior Walls I

- 1 Right-click **Exterior Walls** and choose **Duplicate**.
- 2 In the **Settings** window for **Edge 2D**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study 3/Parametric Solutions 1 (sol6)**.


Time Average I

- 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 I**.
- 4 From the **Parameter selection (alpha0)** list, choose **First**.
- 5 From the **Time selection** list, choose **From list**.
- 6 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 I**.
- 4 From the **Parameter selection (alpha0)** list, choose **From list**.
- 5 In the **Parameter values (alpha0 (deg))** list box, 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 I**.
- 4 From the **Parameter selection (alpha0)** list, choose **From list**.
- 5 In the **Parameter values (alpha0 (deg))** list box, 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 I**.
- 4 From the **Parameter selection (alpha0)** list, choose **From list**.
- 5 In the **Parameter values (alpha0 (deg))** list box, 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.

Lift coefficient stationary Aoa=2

- 1 Right-click **Lift coefficient stationary Aoa=0** and choose **Duplicate**.
- 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 box, select **2**.

Lift coefficient stationary Aoa=4

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

Lift coefficient stationary Aoa=6

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

Lift coefficient transient Aoa=0

- 1 Right-click **Lift coefficient stationary Aoa=6** and choose **Duplicate**.
- 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

Lift coefficient transient Aoa=2

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



Lift coefficient transient Aoa=4

- 1 Right-click **Lift coefficient transient Aoa=2** and choose **Duplicate**.
- 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**.



Lift coefficient transient Aoa=6

- 1 Right-click **Lift coefficient transient Aoa=4** and choose **Duplicate**.
- 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_A0a_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_A0a_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_A0a_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_A0a_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**.

Line 1

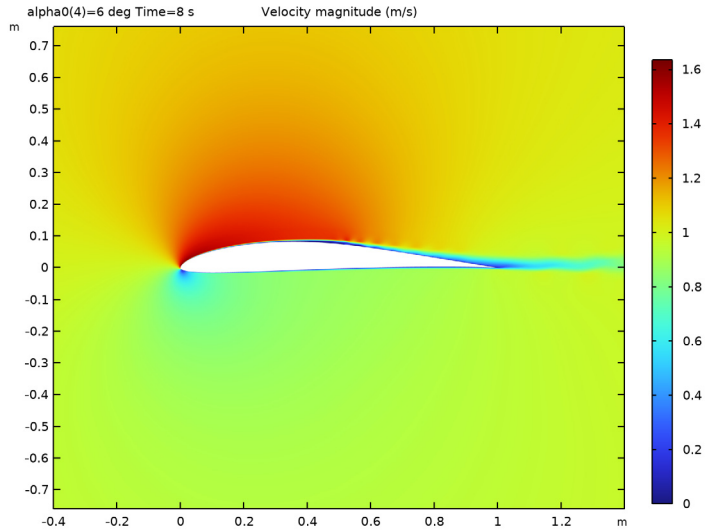
- 1 In the **Model Builder** window, expand the **Wall Resolution (spf)** node, then click **Line 1**.
- 2 In the **Settings** window for **Line**, locate the **Coloring and Style** section.
- 3 In the **Tube radius expression** text field, type 0.01.
- 4 Select the **Radius scale factor** checkbox.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

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


Click the **Zoom In** button in the **Graphics** toolbar a few times and then use the right mouse button to center the airfoil.



Line 1

- 1 In the **Model Builder** window, expand the **Wall Resolution (spf2)** node, then click **Line 1**.
- 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** checkbox. In the associated text field, type 1.
- 5 In the **Wall Resolution (spf2)** toolbar, click  **Plot**.

Aoa=0

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Aoa=0 in the **Label** text field.
- 3 Locate the **Plot Settings** section.
- 4 Select the **y-axis label** checkbox. In the associated text field, type $-C_p$.
- 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 $\alpha = 0^\circ$.

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) > Line 1** 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** checkbox.
- 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** checkbox.
- 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** checkbox.
- 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=2

- 1 Right-click **Aoa=0** and choose **Duplicate**.
- 2 In the **Settings** window for **ID 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 $\alpha = 20^\circ$** .


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 box, 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=4

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

- 2 In the **Settings** window for **ID 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 $\alpha = 4^\circ$.


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 box, 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=6

- 1 In the **Model Builder** window, right-click **Aoa=4** and choose **Duplicate**.
- 2 In the **Settings** window for **ID 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 $\alpha = 6^\circ$.

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 box, 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**.