Turbulent Flow Over a Backward Facing Step

Introduction

The backward facing step has long been a central benchmark case in computational fluid dynamics. The geometry is shown in Figure 1.

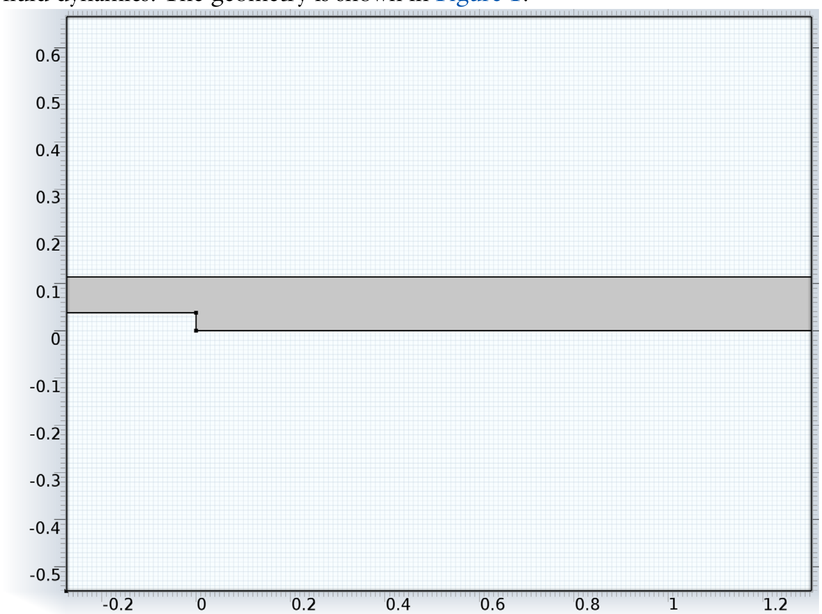

Figure 1: Backstep geometry. Dimensions in SI units.

Fully developed channel flow enters at the domain from the left. When the flow reaches the step, it detaches and a recirculation zone is formed behind the step. Because of the expansion of the channel, the flow slows down and eventually reattaches. The flow field is displayed in Figure 2.

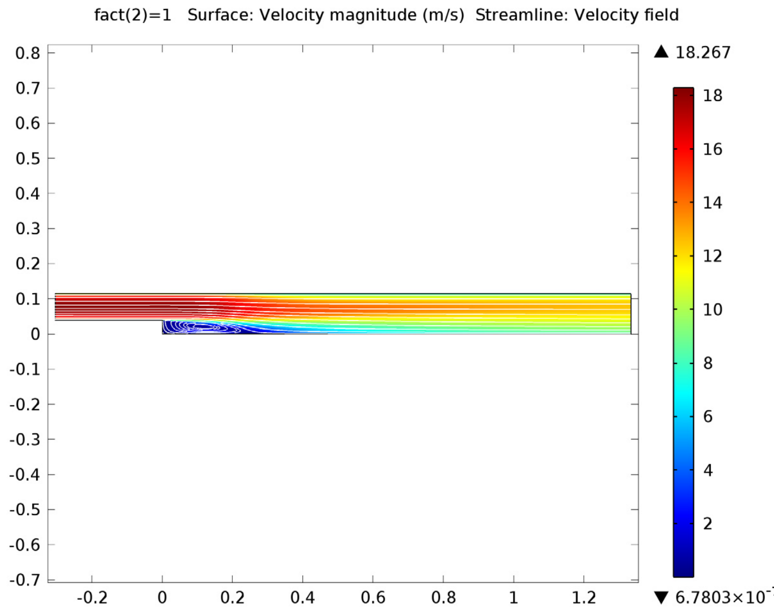

Figure 2: Resulting flow field.

Though seemingly simple, the flow field is a challenge for turbulence models that utilize wall functions. The reason is that wall functions are derived by invoking equilibrium assumptions. Separation and reattachment do not adhere to these assumptions and it must therefore be asserted by numerical experiments that the wall functions can give accurate results even if the underlying theoretical assumptions are not strictly satisfied. The experiment is motivated by the fact that flow with separation and subsequent reattachment are of central importance in many engineering applications.

Model Definition

The model data is taken from Ref. 1. The parameters are given in Table 1. The Reynolds number based on V_{inl} and the step height, S, is $4.8 \cdot 10^4$ and the flow is therefore clearly turbulent.

TABLE 1: MODEL PARAMETERS

PROPERTY	VALUE	DESCRIPTION
S	0.0381 m	Step height
h_c	2·S	Inlet channel height
H	3·S	Outlet channel height
L1	0.3048 m	Inlet channel length
L2	1.3335 m	Outlet channel length
V_{inl}	18.2 m/s	Velocity at centre of upstream channel
ρ	1.23 kg/m^3	Density
μ	$1.79 \cdot 10^{-5}$	Dynamic viscosity

You build the model in two steps:

1 Simulate flow in a long channel of the same height as the inlet to give inlet boundary conditions for the actual geometry.

2 Simulate the flow over the backward facing step using the inlet boundary condition from Step 1.

THE INLET CHANNEL

Ref. 1 suggests to simulate a channel that is $100 \cdot h_c$ in length. Because the channel is symmetric around the midplane, the geometry is taken to be a rectangle with lower left corner at $(x, y) = (0, 0)$ and upper right corner at $(x, y) = (100 \cdot h_c, 0.5 \cdot h_c)$. The upper boundary at $y = 0.5 \cdot h_c$ is a symmetry plane and the lower boundary at $y = 0$ is the wall.

Inlet Boundary Conditions

At the inlet $x = 0$, a plug flow boundary condition with 3% turbulent intensity and a turbulent length scale according to Table 3-4 in the section Theory for the Turbulent Flow User Interfaces in the *CFD Module User's Guide* is prescribed. The inflow velocity cannot be set directly to 18.2 m/s since the resulting centerline velocity at the outlet then becomes too high. While it is possible to set up an ODE that automatically computes the appropriate inlet velocity, it is far easier for small models like this one to find it by trial and error. A few iterations reveal that an inlet velocity of 16.58 m/s gives a centerline value at the outlet very close to 18.2 m/s.

Outlet Boundary Conditions

Outlet boundary conditions can give local artifacts at the outlet. One possible strategy is to elongate the channel and extract data some distance before the outlet. That is however not necessary since fully developed flow in a channel only has a velocity component tangential to the wall, that is normal to the outflow. By prescribing that the outflow must have no tangential component, the outlet artifacts can be removed.

THE BACKWARD FACING STEP

There are two aspects of the backward facing step that need special consideration.

Mesh Generation

It is important to apply a fine enough mesh at the separation point to accurately capture the creation of the shear layer. It must also be remembered that both the flow field and turbulence variables can feature strong gradients close to the walls and that the mesh must be fine enough there to represent these gradients.

Solver Settings

The balance between the turbulence transport equations and the Navier-Stokes equations is rather delicate. If an iteration brings the flow into a state with unphysically large gradients, there is a considerable risk that the simulation will diverge. It is therefore advisable to use the parametric solver to gradually increase the Reynolds number of the flow. The most robust way is to decrease the viscosity which will be done in this model.

Results and Discussion

As shown in Figure 3, the recirculation length normalized by the step height becomes 6.93. Ref. 2 gives an experimental result of 7.1. The result provided by COMSOL is well within the range shown by other investigations (see Ref. 1 and Ref. 3). The separation lengths in Ref. 1 ranges between 6.12 and 7.24. In Ref. 3, recirculation lengths between 5.4 and 7.1 are obtained. Furthermore, Ref. 3 shows that the recirculation length can differ significantly by just changing some implementation details in the wall functions.

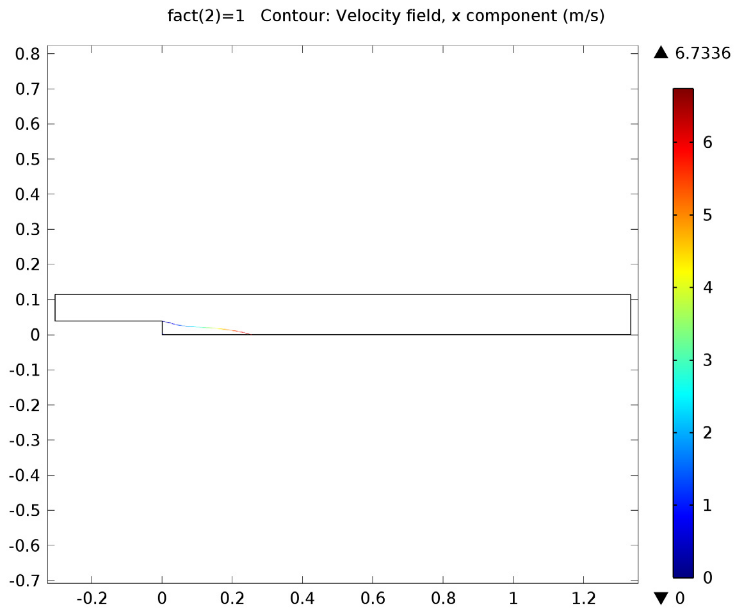

Figure 3: Contour plot of streamwise velocity equal to zero, coloured by x/S where S is the step height.

Finally, note that the recirculation length can shift quite significantly with the mesh resolution. The current result does not shift much if the mesh is refined, but coarser meshes can yield very different recirculation lengths. This emphasizes the need to ensure that the mesh is fine enough.

References

1. *1st NAFEMS Workbook of CFD Examples. Laminar and Turbulent Two-Dimensional Internal Flows.* NAFEMS, 2000.

2. J. Kim, S.J. Kline, and J.P. Johnston, "Investigation of a Reattaching Turbulent Shear Layer: Flow Over a Backward Facing Step," *Transactions of the ASME*, vol. 102, p. 302, 1980.

3. D. Kuzmin, O. Mierka, and S. Turek, "On the Implementation of the k-ε Turbulence Model in Incompressible Flow Solvers Based on a Finite Element

Discretization," *International Journal of Computing Science and Mathematics*, vol. 1, no. 2–4, pp. 193–206, 2007.

Model Library path: `CFD_Module/Single-Phase_Benchmarks/turbulent_backstep`

Modeling Instructions

From the **File** menu, choose **New**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D** button.

3 Click **Next**.

4 In the **Add physics** tree, select **Fluid Flow>Single-Phase Flow>Turbulent Flow>Turbulent Flow, k-ε (spf)**.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

7 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

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

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Description
S	0.0381[m]	Step height
hc	0.0762[m]	Inlet channel height
H	0.1143[m]	Outlet channel height
L1	0.3048[m]	Inlet channel length
L2	1.3335[m]	Outlet channel length
Vinl	16.58[m/s]	Centerline inlet velocity
rhof	1.23[kg/m^3]	Density

Name	Expression	Description
muf	1.79e-5[Pa*s]*fact	Dynamic Viscosity
fact	1.0	Viscosity scaling factor

GEOMETRY 1

Rectangle 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Rectangle**.
2 In the **Rectangle** settings window, locate the **Size** section.
3 In the **Width** edit field, type `100*L1`.
4 In the **Height** edit field, type `hc/2`.
5 Click the **Build All** button.

MATERIALS

Material 1

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.
2 In the **Material** settings window, locate the **Material Contents** section.
3 In the table, enter the following settings:

Property	Name	Value
Density	rho	rhof
Dynamic viscosity	mu	muf

Turbulent Flow, k-ε

Inlet 1

1 In the **Model Builder** window, under **Model 1** right-click **Turbulent Flow, k-ε** and choose **Inlet**.
2 Select Boundary 1 only.
3 In the **Inlet** settings window, locate the **Velocity** section.
4 In the U_0 edit field, type `Vinl`.
5 Locate the **Boundary Condition** section. In the L_T edit field, type `0.07*hc`.
6 In the I_T edit field, type `0.03`.

Symmetry 1

1 In the **Model Builder** window, right-click **Turbulent Flow, k-ε** and choose **Symmetry**.

2 Select Boundary 3 only.

Boundary Stress 1

1 Right-click **Turbulent Flow, k-ε** and choose **Boundary Stress**.

2 In the **Boundary Stress** settings window, locate the **Boundary Condition** section.

3 From the **Boundary condition** list, choose **Normal stress, normal flow**.

4 Select Boundary 4 only.

MESH 1

Mapped 1

In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Mapped**.

Distribution 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 1 and 4 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 From the **Distribution method** list, choose **Geometric sequence**.

6 In the **Element ratio** edit field, type 10.

7 In the **Number of elements** edit field, type 40.

Distribution 2

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

2 Select Boundary 2 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** edit field, type 250.

6 In the **Element ratio** edit field, type 2.

7 Select the **Reverse direction** check box.

Distribution 3

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

2 In the **Distribution** settings window, locate the **Distribution** section.

3 From the **Distribution properties** list, choose **Predefined distribution type**.

4 Select Boundary 3 only.

5 In the **Number of elements** edit field, type `250`.

6 In the **Element ratio** edit field, type `2`.

7 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Check that the flow is fully developed by plotting `muT` along the centerline.

1D Plot Group 4

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 Right-click **1D Plot Group 4** and choose **Line Graph**.

3 Select Boundary 3 only.

This is the top surface.

4 In the **Line Graph** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Turbulent Flow, k-ε>Turbulent dynamic viscosity (spf.muT)**.

5 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Rename**.

6 Go to the **Rename 1D Plot Group** dialog box and type `Turbulent viscosity` in the **New name** edit field.

7 Click **OK**.

8 Right-click **1D Plot Group 4** and choose **Plot**.

As can be seen in the resulting plot (Figure 4), the turbulent viscosity has obtained a constant value well before the outlet

.

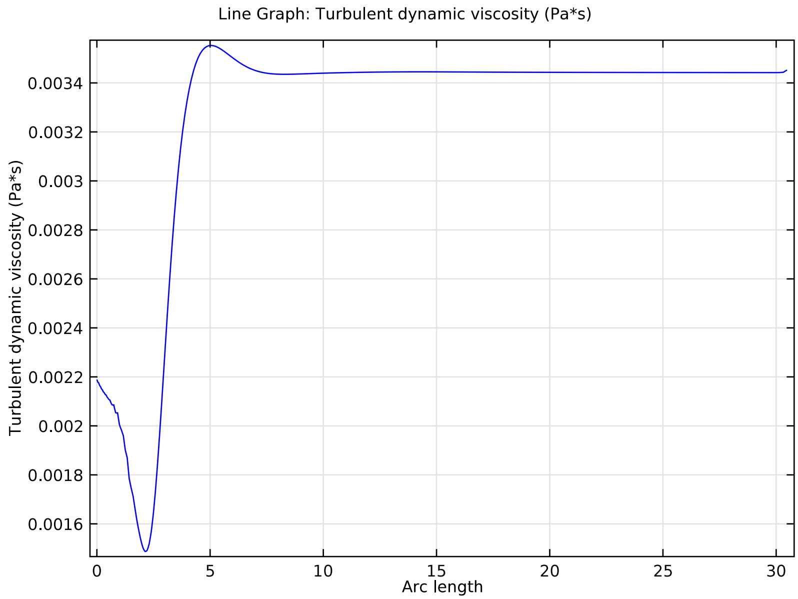

Figure 4: Turbulent viscosity along the centerline of the inlet channel.

With the initial simulation step completed, create the backstep model.

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Model**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D** button.

3 Click **Next**.

4 In the **Add physics** tree, select **Recently Used>Turbulent Flow, k-ε (spf)**.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Stationary**.

7 Find the **Selected physics** subsection. In the table, enter the following settings:

Physics	Solve for
Turbulent Flow, k-ε (spf)	×

8 Click **Finish**.

GEOMETRY 2

Rectangle 1

1 In the **Model Builder** window, under **Model 2** right-click **Geometry 2** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `L1+L2`.

4 In the **Height** edit field, type `hc`.

5 Locate the **Position** section. In the **x** edit field, type `-L1`.

6 In the **y** edit field, type `S`.

Point 1

1 In the **Model Builder** window, right-click **Geometry 2** and choose **Point**.

2 In the **Point** settings window, locate the **Point** section.

3 In the **x** edit field, type `-L1`.

4 In the **y** edit field, type `S+hc/2`.

Union 1

1 Right-click **Geometry 2** and choose **Boolean Operations>Union**.

2 Select the objects **r1** and **pt1** only.

Rectangle 2

1 Right-click **Geometry 2** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `L2`.

4 In the **Height** edit field, type `S`.

5 Click the **Build Selected** button.

Union 2

1 Right-click **Geometry 2** and choose **Boolean Operations>Union**.

2 Select the objects **uni1** and **r2** only.

Mesh Control Edges 1

1 Right-click **Geometry 2** and choose **Virtual Operations>Mesh Control Edges**.

2 On the object **fin**, select Boundary 7 only.

3 Right-click **Geometry 2** and choose **Build All**.

4 Click the **Zoom Extents** button on the Graphics toolbar.

DEFINITIONS

Linear Extrusion 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Model Couplings>Linear Extrusion**.

2 In the **Linear Extrusion** settings window, locate the **Source Selection** section.

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

4 Select Boundary 4 only.

5 In the **Linear Extrusion** settings window, locate the **Source Vertices** section.

6 Under **Source vertex 1**, click **Activate Selection**.

7 Select Point 3 only.

8 In the **Linear Extrusion** settings window, locate the **Source Vertices** section.

9 Under **Source vertex 2**, click **Activate Selection**.

10 Select Point 4 only.

11 In the **Linear Extrusion** settings window, click to expand the **Destination** section.

12 From the **Destination geometry** list, choose **Geometry 2**.

13 Locate the **Destination Vertices** section. Under **Destination vertex 1**, click **Activate Selection**.

14 Select Point 1 only.

15 In the **Linear Extrusion** settings window, locate the **Destination Vertices** section.

16 Under **Destination vertex 2**, click **Activate Selection**.

17 Select Point 2 only.

Linear Extrusion 2

1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Linear Extrusion**.

2 In the **Linear Extrusion** settings window, locate the **Source Selection** section.

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

4 Select Boundary 4 only.

5 In the **Linear Extrusion** settings window, locate the **Source Vertices** section.

6 Under **Source vertex 1**, click **Activate Selection**.

7 Select Point 3 only.

8 In the **Linear Extrusion** settings window, locate the **Source Vertices** section.

9 Under **Source vertex 2**, click **Activate Selection**.

10 Select Point 4 only.

11 In the **Linear Extrusion** settings window, locate the **Destination** section.

12 From the **Destination geometry** list, choose **Geometry 2**.

13 Locate the **Destination Vertices** section. Under **Destination vertex 1**, click **Activate Selection**.

14 Select Point 3 only.

15 In the **Linear Extrusion** settings window, locate the **Destination Vertices** section.

16 Under **Destination vertex 2**, click **Activate Selection**.

17 Select Point 2 only.

MATERIALS

Material 2

1 In the **Model Builder** window, under **Model 2** right-click **Materials** and choose **Material**.

2 In the **Material** settings window, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value
Density	rho	`rhof`
Dynamic viscosity	mu	`muf`

Turbulent Flow, k-ε 2

Inlet 1

1 In the **Model Builder** window, under **Model 2** right-click **Turbulent Flow, k-ε 2** and choose **Inlet**.

2 Select Boundary 1 only.

3 In the **Inlet** settings window, locate the **Velocity** section.

4 Click the **Velocity field** button.

5 In the $\mathbf{u}_0$ table, enter the following settings:

`mod1.linext1(mod1.u)`	x
`0`	y

6 Locate the **Boundary Condition** section. Click the **Specify turbulence variables** button.

7 In the k_0 edit field, type `mod1.linext1(mod1.k)`.

8 In the ε_0 edit field, type `mod1.linext1(mod1.ep)`.

Inlet 2

1 In the **Model Builder** window, right-click **Turbulent Flow, k-ε 2** and choose **Inlet**.

2 Select Boundary 3 only.

3 In the **Inlet** settings window, locate the **Velocity** section.

4 Click the **Velocity field** button.

5 In the $\mathbf{u}_0$ table, enter the following settings:

`mod1.linext2(mod1.u)`	x
`0`	y

6 Locate the **Boundary Condition** section. Click the **Specify turbulence variables** button.

7 In the k_0 edit field, type `mod1.linext2(mod1.k)`.

8 In the ε_0 edit field, type `mod1.linext2(mod1.ep)`.

Outlet 1

1 Right-click **Turbulent Flow, k-ε 2** and choose **Outlet**.

2 In the **Outlet** settings window, locate the **Boundary Condition** section.

3 From the **Boundary condition** list, choose **Pressure**.

4 Select Boundary 7 only.

MESH 2

1 In the **Model Builder** window, under **Model 2** click **Mesh 2**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Coarse**.

Size 1

1 Right-click **Model 2>Mesh 2** and choose **Edit Physics-Induced Sequence**.

2 In the **Model Builder** window, under **Model 2>Mesh 2** right-click **Size 1** and choose **Build Selected**.

Size 2

1 Right-click **Mesh 2** and choose **Size**.

2 In the **Size** settings window, locate the **Geometric Entity Selection** section.

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

4 Select Boundary 9 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element growth rate** check box.

7 In the associated edit field, type `1.03`.

Size 3

1 Right-click **Mesh 2** and choose **Size**.

2 In the **Size** settings window, locate the **Geometric Entity Selection** section.

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

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 Select Point 5 only.

7 In the associated edit field, type `5e-4`.

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Model 2>Mesh 2>Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

2 In the **Boundary Layer Properties** settings window, locate the **Boundary Layer Properties** section.

3 In the **Thickness adjustment factor** edit field, type `2`.

4 In the **Number of boundary layers** edit field, type `6`.

5 Click the **Build All** button.

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 2** node, then click **Step 1: Stationary**.

2 In the **Stationary** settings window, click to expand the **Study Extensions** section.

3 Select the **Continuation** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Continuation parameter	Parameter value list
fact	5 1

6 Click to expand the **Values of Dependent Variables** section. Select the **Values of variables not solved for** check box.

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

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

9 In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

RESULTS

Check that the wall lift-off is `11.06` almost everywhere by selecting the **Wall Resultion (spf2)** plot group.

Wall Resolution (spf2)

Next, reproduce the flow-field plot with the following steps:

Velocity (spf2)

1 In the **Model Builder** window, under **Results** right-click **Velocity (spf2)** and choose **Streamline**.

2 In the **Streamline** settings window, locate the **Streamline Positioning** section.

3 From the **Positioning** list, choose **Uniform density**.

4 In the **Separating distance** edit field, type `0.007`.

5 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

6 Click the **Plot** button.

Finally, visualize the recirculation length.

2D Plot Group 8

1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.

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

3 From the **Data set** list, choose **Solution 3**.

4 Right-click **Results>2D Plot Group 8** and choose **Contour**.

5 In the **Contour** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Turbulent Flow, k-ε 2>Velocity field>Velocity field, x component (u2)**.

6 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

7 Right-click **Results>2D Plot Group 8>Contour 1** and choose **Color Expression**.

8 In the **Color Expression** settings window, locate the **Expression** section.

9 In the **Expression** edit field, type `x/S`.

10 In the **Model Builder** window, right-click **2D Plot Group 8** and choose **Rename**.

11 Go to the **Rename 2D Plot Group** dialog box and type `Recirculation length` in the **New name** edit field.

12 Click **OK**.

13 In the **2D Plot Group** settings window, locate the **Data** section.

14 From the **Data set** list, choose **Solution 3**.

15 Click the **Plot** button.