Created in COMSOL Multiphysics 5.6

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.

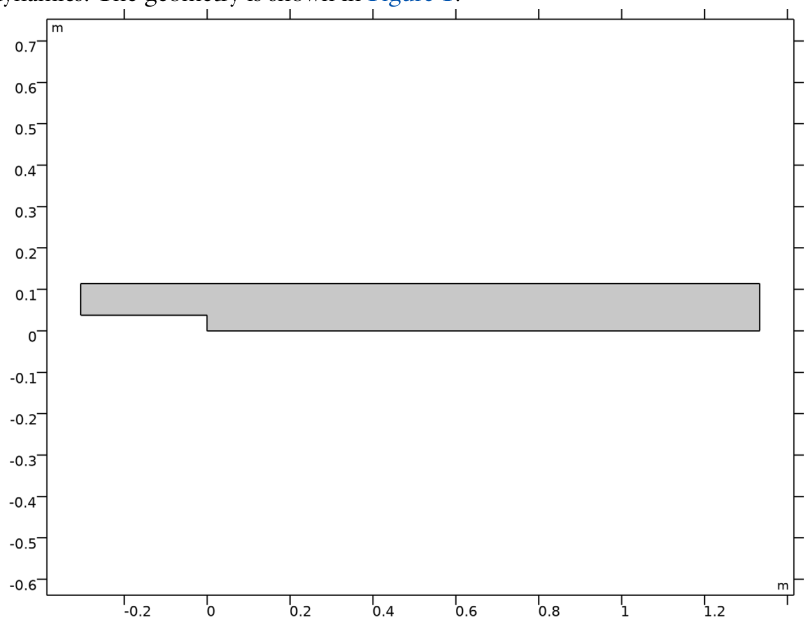


Figure 1: Backstep geometry. Dimensions in SI units.

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.

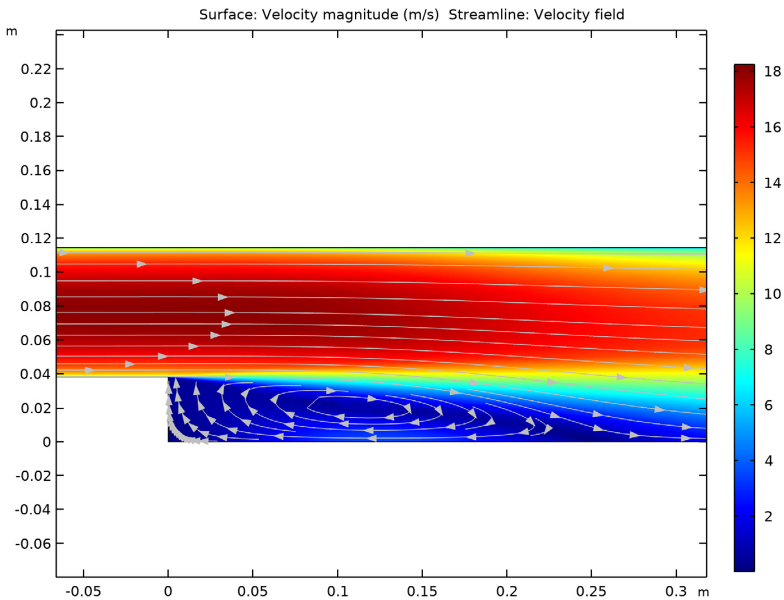


Figure 2: Resulting flow field.

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 center of upstream channel
ρ	1.23 kg/m^3	Density
μ	$1.79 \cdot 10^{-5}$	Dynamic viscosity

Results and Discussion

As shown in Figure 3, the recirculation length normalized by the step height becomes 6.64. Ref. 2 gives an experimental result of 7.1. The result provided by COMSOL Multiphysics 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.

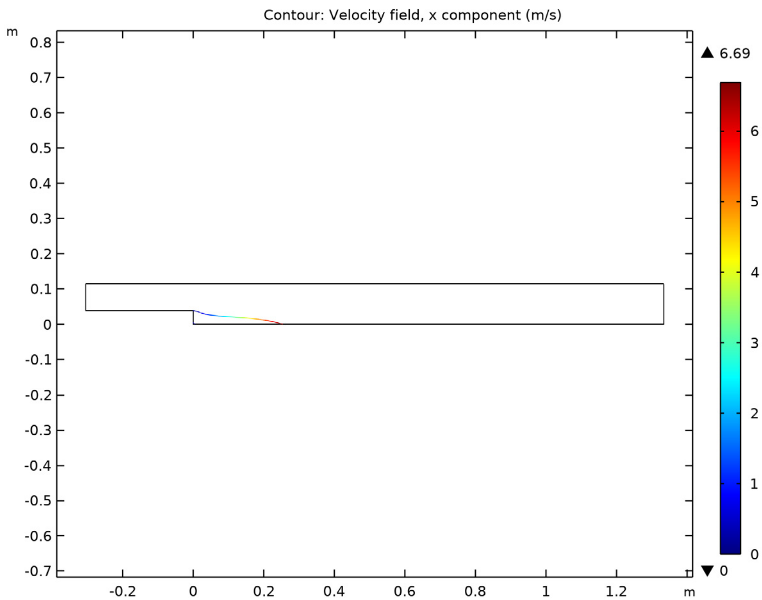


Figure 3: Contour plot of streamwise velocity equal to zero, colored by x/S *where* S *is the step height.*

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”, *Int’l J Computing Science and Mathematics*, vol. 1, no. 2–4, pp. 193–206, 2007.

Notes About the COMSOL Implementation

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.

Inlet Boundary Conditions

To simulate fully developed channel flow that enters the domain, choose Fully developed flow option in the Inlet boundary condition. To achieve centerline velocity 18.2 m/s, set up an ODE that automatically computes average inlet velocity U_{av}.

Application Library path: `CFD_Module/Verification_Examples/turbulent_backstep`

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>Turbulent Flow>Turbulent Flow, k-ε (spf)**.

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
S	0.0381[m]	0.0381 m	Step height
hc	0.0762[m]	0.0762 m	Inlet channel height
H	0.1143[m]	0.1143 m	Outlet channel height
L1	0.3048[m]	0.3048 m	Inlet channel length
L2	1.3335[m]	1.3335 m	Outlet channel length
Vinl	18.2[m/s]	18.2 m/s	Centerline inlet velocity
rhof	1.23[kg/m^3]	1.23 kg/m³	Density
muf	1.79e-5[Pa*s]	1.79E-5 Pa·s	Dynamic viscosity

GEOMETRY 1

Rectangle 1 (r1)

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

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

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

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

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

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

Rectangle 2 (r2)

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 `L2`.

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

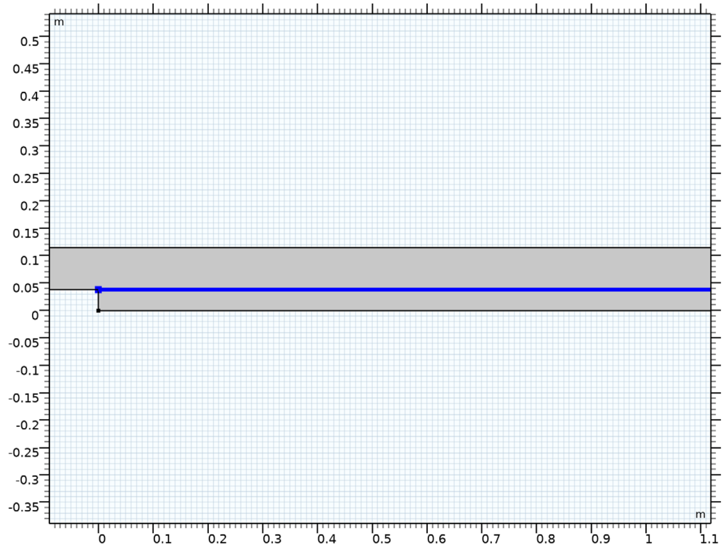
5 Click **Build Selected**.

Mesh Control Edges 1 (mce1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Edges**.

2 On the object **fin**, select Boundary 6 only.

It might be easier to select the correct boundary by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)



3 In the **Geometry** toolbar, click **Build All**.

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

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	`rhof`	kg/m³	Basic
Dynamic viscosity	mu	`muf`	Pa·s	Basic

TURBULENT FLOW, K-ε (SPF)

Inlet 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Turbulent Flow, k-ε (spf)** and choose **Inlet**.

2 Select Boundary 1 only.

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

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

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

The average inlet velocity U_{av} will be defined later.

Outlet 1

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

2 Select Boundary 6 only.

3 In the **Settings** window for **Outlet**, locate the **Pressure Conditions** section.

4 Select the **Normal flow** check box.

Define the probe to monitor the centerline velocity.

DEFINITIONS

Domain Point Probe 1

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Point Probe**.

2 In the **Settings** window for **Domain Point Probe**, locate the **Point Selection** section.

3 In row **Coordinates**, set **x** to `-L1`.

4 In row **Coordinates**, set **y** to `S+hc/2`.

Point Probe Expression 1 (ppb1)

1 In the **Model Builder** window, expand the **Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.

2 In the **Settings** window for **Point Probe Expression**, type `Ucl` in the **Variable name** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `u`.

Define the equation to calculate average inlet velocity.

4 Click the **Show More Options** button in the **Model Builder** toolbar.

5 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

6 Click **OK**.

TURBULENT FLOW, K-ε (SPF)

Global Equations 1

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

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

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
Uav	Ucl-Vinl	Vinl	0	

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `velocity` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Velocity (m/s)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, select **General>Velocity (m/s)** in the tree.

12 Click **OK**.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

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

Size

Right-click **Component 1 (comp1)>Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size 1

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

Size 2

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

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

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

4 Select Boundary 8 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 text field, type `1.03`.

Size 3

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

2 In the **Settings** window for **Size**, 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 4 only.

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

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Mesh 1>Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

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

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

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

5 Click **Build All**.

STUDY 1

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

RESULTS

Wall Resolution (spf)

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

Next, reproduce the flow-field plot (Figure 2) with the following steps:

Streamline 1

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

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

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

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

5 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **Gray**.

6 From the **Type** list, choose **Arrow**.

7 From the **Arrow distribution** list, choose **Equal time**.

8 Select the **Number of arrows** check box.

9 In the associated text field, type `150`.

10 Select the **Scale factor** check box.

11 In the associated text field, type `0.0012`.

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

Next, add and set up a dedicated view that zooms in on the recirculation zone.

View 2D 3

1 In the **Model Builder** window, right-click **Views** and choose **View 2D**.

2 Use the mouse buttons to zoom in to get a closer view.

3 In the **Model Builder** window, click **View 2D 3**.

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

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

Velocity (spf)

1 In the **Model Builder** window, click **Velocity (spf)**.

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

3 From the **View** list, choose **View 2D 3** to apply the view you just created.

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

Finally, visualize the recirculation length (Figure 3).

2D Plot Group 5

In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

Contour 1

1 Right-click **2D Plot Group 5** and choose **Contour**.

2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Turbulent Flow, k-ε>Velocity and pressure>Velocity field - m/s>u - Velocity field, x component**.

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

Color Expression 1

1 Right-click **Contour 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Expression** section.

3 In the **Expression** text field, type `x/S`.

Recirculation length

1 In the **Model Builder** window, click **2D Plot Group 5**.

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

3 Select the **Show maximum and minimum values** check box.

4 Right-click **2D Plot Group 5** and choose **Rename**.

5 In the **Rename 2D Plot Group** dialog box, type `Recirculation length` in the **New label** text field.

6 Click **OK**.

7 In the **Recirculation length** toolbar, click **Plot**.