

# Turbulent Flow over a Backward-Facing Step

# Introduction





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  $U_{\rm av}$  and the step height, S, is 47625 and the flow is therefore clearly turbulent.

Property	Value	Description		
S	0.0381 m	Step height		
h <sub>c</sub>	2·S	Inlet channel height		
Н	3·S	Outlet channel height		
LI	0.3048 m	Inlet channel length		
L2	1.3335 m	Outlet channel length		
ρ	1.23 kg/m <sup>3</sup>	Density		
μ	1.79·10 <sup>-5</sup>	Dynamic viscosity		
Re	47625	Reynolds number		
U <sub>av</sub>	$Re{\cdot}\mu/\rho/S$	Mean inlet velocity		

TABLE I: MODEL PARAMETERS.

# Results and Discussion

As shown in Figure 3, the recirculation length normalized by the step height becomes close to 6.7. 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.

# 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, enforcing an average inlet velocity  $U_{av}$  based on the Reynolds number.

# References

1. Ist 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- $\varepsilon$  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.

**Application Library path:** Heat\_Transfer\_Module/Verification\_Examples/ turbulent\_backstep

# Modeling Instructions

From the File menu, choose New.

#### NEW

In the New window, click 🔗 Model Wizard.

#### MODEL WIZARD

- I In the Model Wizard window, click 🍳 2D.
- 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 M Done.

# GLOBAL DEFINITIONS

# Parameters 1

I 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	
S	0.0381[m]	0.0381 m	Step height	
hc	0.0762[m]	0.0762 m	Inlet channel height	
Н	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	
rhof	1.23[kg/m^3]	1.23 kg/m <sup>3</sup>	Density	
muf	1.79e-5[Pa*s]	1.79E-5 Pa·s	Dynamic viscosity	
Re	47625	47625	Reynolds number	
Uav	Re*muf/rhof/S	18.191 m/s	Mean inlet velocity	

#### GEOMETRY I

Rectangle 1 (r1)

- I 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)

- I 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 틤 Build Selected.

# Mesh Control Edges 1 (mcel)

- I 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.)



**<sup>4</sup>** Click the **F Zoom Extents** button in the **Graphics** toolbar.

# MATERIALS

Material I (mat1)

- I In the Model Builder window, under Component I (compl) 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, $K-\epsilon$ (SPF)

#### Inlet 1

- I In the Model Builder window, under Component I (compl) right-click Turbulent Flow, k- $\epsilon$  (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.

# Outlet I

- I 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.

# MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 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 I (compl)>Mesh I and choose Edit Physics-Induced Sequence.

#### Size 1

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

#### Size 2

- I In the Model Builder window, right-click Mesh I 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.
- 7 Select the Maximum element growth rate check box. In the associated text field, type 1.03.

Size 3

- I Right-click Mesh I 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. In the associated text field, type 5e-4.

Boundary Layer Properties 1

- In the Model Builder window, expand the Component I (compl)>Mesh I>
   Boundary Layers I node, then click Boundary Layer Properties I.
- 2 In the Settings window for Boundary Layer Properties, locate the Layers section.
- **3** In the **Thickness adjustment factor** text field, type **2**.
- 4 In the Number of layers text field, type 6.
- 5 Click 📗 Build All.

#### STUDY I

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

#### RESULTS

#### Wall Resolution (spf)

In this case the wall lift-off is limited from below by 11.06 which can be seen by selecting the **Wall Resolution (spf)** plot group.

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

Streamline 1

- I 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 0.005.
- 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. In the associated text field, type 150.

9 Select the Scale factor check box. In the associated text field, type 0.0008.

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

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

View 2D 3

- I In the Model Builder window, under Results 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)

- I In the Model Builder window, under Results 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 4

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

#### Contour I

- I Right-click 2D Plot Group 4 and choose Contour.
- In the Settings window for Contour, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>Turbulent Flow, k-&>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

- I Right-click Contour I 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

- I In the Model Builder window, under Results click 2D Plot Group 4.
- 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** In the **Label** text field, type Recirculation length.

**5** In the **Recirculation length** toolbar, click **O** Plot.