

## Example: Steady Incompressible Flow

### Introduction

This model examines the physics of plane, incompressible, and steady flow: flow over a backward step in the absence of external forces. This is a common benchmark problem in CFD. There is no known exact solution, but experimental data has been published (see [Ref. 1](#)) making it possible to check the accuracy of the FEM solution. The model includes analyses using both regular triangular meshes and mapped meshes, comparing the solutions for various mesh densities.

### Model Definition

Fluid enters from the left side with a parabolic velocity profile, passes over a step, and leaves through the right boundary ([Figure 6-1](#) shows the model geometry).

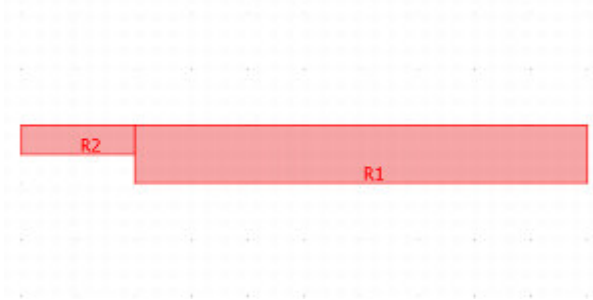


Figure 6-1: The backstep geometry.

The model computes the fluid's velocity components  $\mathbf{u} = (u, v)$  in the  $x$ - and  $y$ -directions and its pressure  $p$  in the region defined by the geometry in the preceding figure. The PDE model for this application uses the stationary incompressible Navier-Stokes equations

$$\begin{cases} -\eta \nabla^2 \mathbf{u} + \rho (\mathbf{u} \cdot \nabla) \mathbf{u} + \nabla p = \mathbf{F} \\ \nabla \cdot \mathbf{u} = 0 \end{cases}$$

where the quantities are (using SI units):

- Dynamic viscosity,  $\eta = 1.79 \cdot 10^{-5}$  kg/ms
- Density,  $\rho = 1.23$  kg/m<sup>3</sup>
- A force field,  $\mathbf{F}$ , absent in this model

The first equation is the balance of momentum from Newton's second law. The other relationship is the equation of continuity, where zero on the right-hand side states that the fluid is incompressible.

The shape of the flow pattern depends only on the Reynolds number.

In this model, you choose boundary conditions so that the average velocity at the inlet is  $V_{mean} = 0.544$  m/s. To obtain a corresponding parabolic velocity profile, set  $(u, v) = (6V_{mean}s(1-s), 0)$ , where  $s$  is a boundary parameterization variable that runs from 0 to 1 along the boundary. The fluid is always stationary at the walls, so  $(u, v) = (0, 0)$  is the appropriate boundary condition. At the exit boundary, assume a constant static pressure  $p = 0$ .

For such a fluid flow you can expect a velocity field with a boundary layer of thickness  $\approx 1/\sqrt{Re}$  at the walls. To resolve this steep solution gradient you need a few rows of elements across the layer. For a flow with a large Reynolds number, elements in the interior of the channel can be much larger than those near the walls.

### USING THE INCOMPRESSIBLE NAVIER-STOKES APPLICATION MODE

Start by setting up a model and solve this problem on a fixed isotropic mesh. The given input data correspond to a Reynolds number of

$$Re = \frac{0,544 \cdot 2 \cdot 0,0052 \cdot 1,23}{1,79 \cdot 10^{-5}} \approx 389$$

Even though you are working with a steady flow model, it needs initial conditions because the incompressible Navier-Stokes equations are nonlinear. To achieve a numerical solution, the nonlinear solver solves the equations iteratively.

## Results

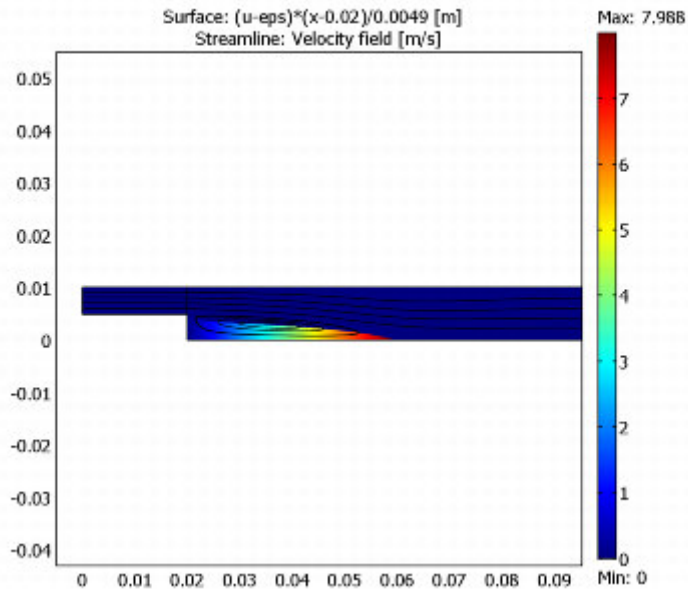


Figure 6-2: A streamline plot of the velocity field.

Figure 6-2 shows the velocity field in a streamline plot. Behind the step you can identify a recirculation region that expands with increasing  $Re$ . The distance from the step to the *stagnation point*, where the flow reattaches to the lower wall, is the *reattachment length*. A dimensionless number quantifying the recirculation region is the reattachment length divided by the step height. According to experimental data, this quotient is approximately 7.93 at a Reynolds number of 389.

For a high enough  $Re$ , the eddy crosses the outflow boundary, which means that there is an inflow through the assumed outflow boundary. This finding calls for the inclusion of more of the downstream region in the computational domain to get a reliable solution. That fact explains why the computational domain in this model is slightly longer than the one in [Ref. 3](#).

The Incompressible Navier-Stokes application mode uses Lagrange p2-p1 elements to stabilize the pressure. Thus 2nd-order Lagrange elements model the velocity components while linear elements model the pressure. The default element settings in this application mode always provides one order higher Lagrange elements for the velocity components than for the pressure. Stabilizing the solution using streamline diffusion is not recommended for the Navier-Stokes equations except at higher Reynolds numbers.

Four mesh cases were considered for this model: a homogeneous structured mesh, a homogeneous unstructured mesh, and two inhomogeneous mesh cases, where local mesh refinements provide improved accuracy around the point where the step is taken. The following plots show examples of the mesh cases:

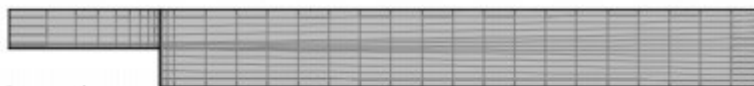


Case 1: Homogeneous structured mesh

Case 2: Homogeneous unstructured mesh



Case 3: Structured mesh



Case 4: Unstructured mesh



To compare the results using these mesh cases, we solved the model for varying mesh densities for all four mesh cases. The graph in [Figure 6-3](#) shows the resulting reattachment length divided by the step height as a function of the degrees of freedom for the different mesh cases:

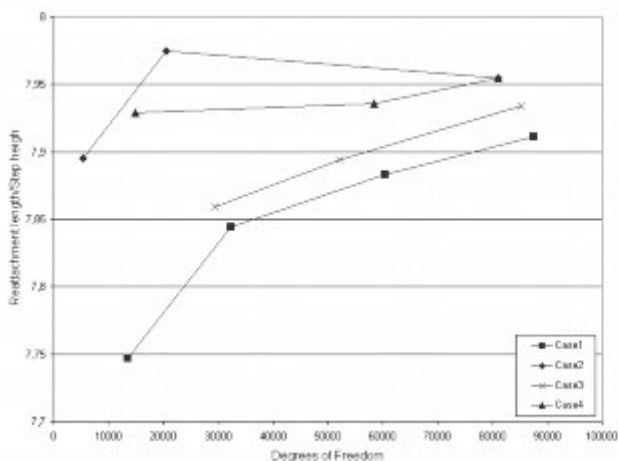


Figure 6-3: Reattachment length divided by step height as functions of the degrees of freedom for Case 1 (squares), Case 2 (diamonds), Case 3 (x-marks), and Case 4 (triangles).

All solutions give reasonable results. Comparing results for structured and unstructured meshes and different numbers of degrees of freedom (DOFs), it is evident that the unstructured mesh cases give more accurate and more stable results already for a lower number of DOFs. However, in the limit of an infinite number of DOFs, the results for all mesh cases converge to the experimental result of 7.93.

The number of mesh elements is related to the number of DOFs. However, the relation is not the same for structured and unstructured meshes. The same number of mesh elements create more DOFs for a structured mesh than for an unstructured mesh, simply because the rectangular shape of the mesh elements.

For the same number of DOFs the solution time is longer for a structured mesh than for an unstructured mesh. This is due to the structured mesh having a stronger coupled system, which results in system matrices that are less sparse.

---

**Model Library Path:** COMSOL\_Multiphysics/Fluid\_Dynamics/backstep

---



---

**Model Library Path:** COMSOL\_Multiphysics/Fluid\_Dynamics/backstep\_quad

---

## Modeling Using the Graphical User Interface

---

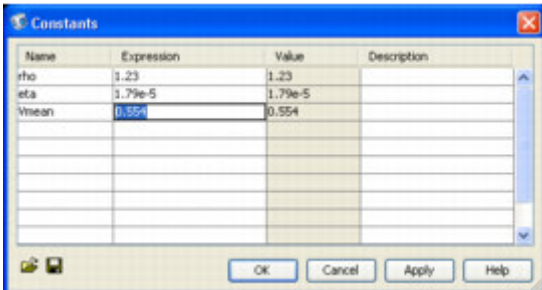
### MODEL NAVIGATOR

- 1 Go to the **Model Navigator** and select **2D** in the **Space dimension** list.
- 2 In the list of application modes open the **COMSOL Multiphysics>Fluid Dynamics** folder and then the **Incompressible Navier-Stokes** folder. Select **Steady-state analysis**.
- 3 Make sure that **Lagrange - p2-p1** is the element type in the **Element** list.
- 4 Click **OK**.

## OPTIONS AND SETTINGS

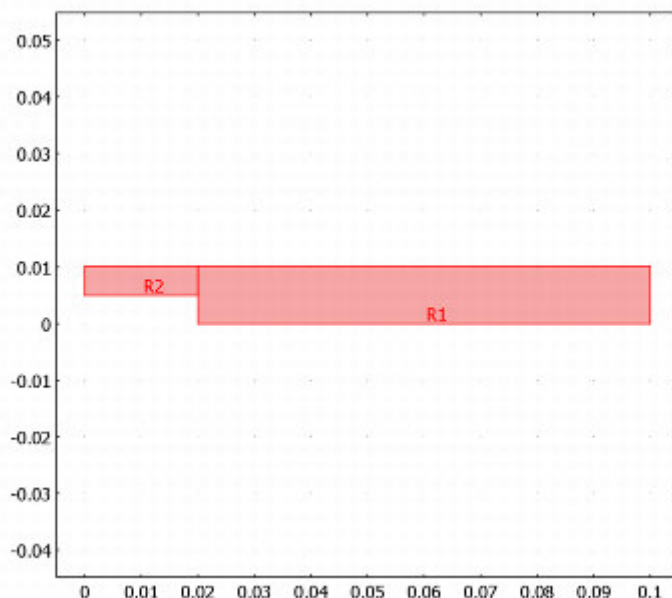
- 1 To parameterize the model, go to the **Options** menu and choose **Constants**.
- 2 Make the following entries in the **Constants** dialog box to represent the fluid properties and velocity:

NAME	EXPRESSION
rho	1.23
eta	1.79e-5



## GEOMETRY MODELING

- 1 Shift-click the **Rectangle/Square** button to specify a rectangle.
- 2 In the **Rectangle** dialog box, type 0.08 in the **Width** edit field and 0.0101 in the **Height** edit field. Let the **Base Position** be **Corner** and enter 0.02 in the **x** edit field.
- 3 Click **OK**.
- 4 Shift-click the **Rectangle/Square** button to specify another rectangle.
- 5 In the **Rectangle** dialog box, type 0.02 in the **Width** edit field and 0.0052 in the **Height** edit field. Let the **Base Position** be **Corner** and enter 0.0049 in the **y** edit field.
- 6 Click **OK**.
- 7 Click the **Zoom Extents** button.



## PHYSICS SETTINGS

### *Subdomain Settings*

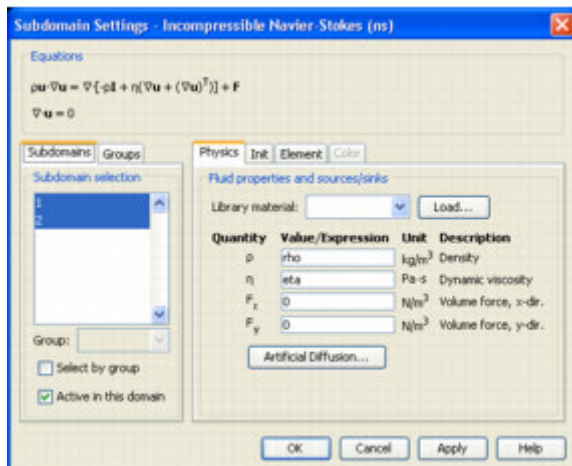
This model uses actual fluid properties specified in SI units. With appropriate scaling it is possible to set the fluid density  $\rho = 1$  and viscosity  $\eta = 1/Re$ . If you model in this way, the Reynolds number contains information about fluid density together with details on length scale, velocity scale, and viscosity. For

such a model, enter  $1/Re$  in the **Dynamic viscosity** edit field in the **Subdomain Settings** dialog box. Be aware, however, that you must reverse the scaling before interpreting the results quantitatively.

Enter these quantities:

SUBDOMAIN	1
$\rho$	rho
$\eta$	eta
$V_{mean}$	0.544

- 1 From the **Physics** menu, choose **Subdomain Settings**.
- 2 In the **Subdomain Settings** dialog box select subdomains 1 and 2 from the **Subdomain selection** list.
- 3 Enter rho in the **Density** edit field.
- 4 Enter eta in the **Dynamic viscosity** edit field.

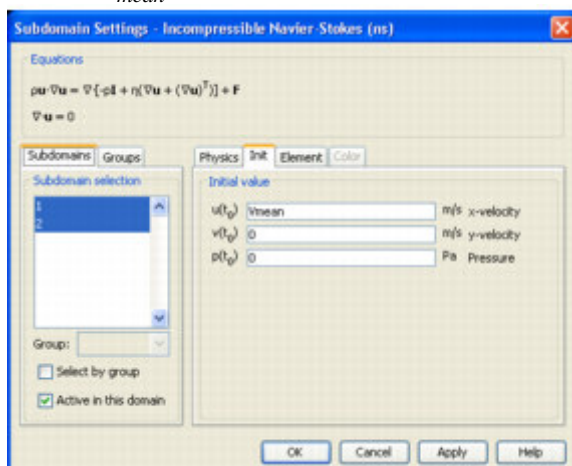


The nonlinear solver also requires an initial value. With a low Reynolds number a simple constant expression is sufficient.

- 5 With both subdomains still selected, click the **Init** tab.
- 6 Set initial conditions according to the following table:

SUBDOMAIN	1 AND 2
$u(t_0)$	$V_{mean}$
$v(t_0)$	0
$p(t_0)$	0

- 7 Enter the initial value  $V_{mean}$  in the edit field for  $u(t_0)$ . This value corresponds to a uniform velocity field of  $V_{mean}$  throughout the domain.



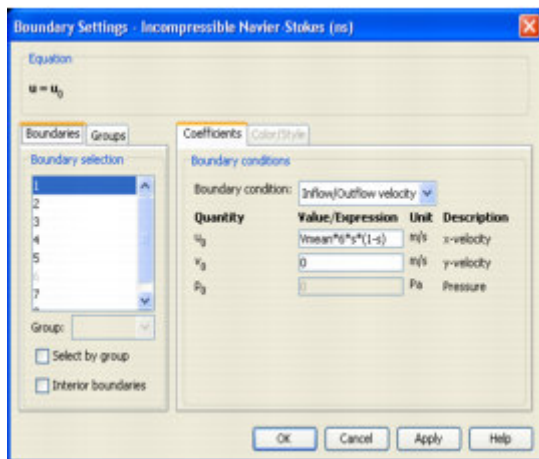
8 Click **OK**.

## Boundary Conditions

- 1 From the **Physics** menu, choose **Boundary Settings**.
- 2 In the **Boundary Conditions** dialog box enter the following boundary coefficients:

BOUNDARY	1	2-5, 7	8
Type	Inflow/Outflow velocity	No-slip	Outflow/Pressure
$u_0$	$v_{\text{mean}} * 6 * s * (1 - s)$		
$v_0$	0		
$p$			0

- 3 Select boundary 1 in the **Boundary selection** list.
- 4 Select **Inflow/Outflow velocity** in the **Boundary condition** list.
- 5 Enter  $v_{\text{mean}} * 6 * s * (1 - s)$  in the  $v_0$  edit field.



- 6 Select boundary 8 in the **Boundary selection** list.
- 7 Select **Outflow/Pressure** in the **Boundary condition** list to fix the pressure and also to specify that there should be no viscous stresses on the fluid at the outlet.
- 8 Keep the default values for boundaries 2-5 and 7.
- 9 Click **OK**.

## MESH GENERATION

Because the Navier-Stokes equations are computationally difficult, it is important to use an appropriate mesh. If the mesh is too coarse, the solution might not converge at all or errors might be large. Conversely, if the mesh is too fine, the solution time for the nonlinear system of equations might be unnecessarily long. In this case, solve the model with two different types of mesh, first using a mapped mesh and later using a conventional triangular mesh.

### Mesh Case 1—Mapped mesh

- 1 Go to the **Mesh** menu and choose **Mapped Mesh Parameters**.
- 2 In the **Mapped Mesh Parameters** dialog box click the **Boundary** tab.
- 3 Select the **Constrained edge element distribution** check box.
- 4 Enter the following **Number of mesh elements**.

BOUNDARY	NUMBER OF EDGE ELEMENTS
1,2,3,4,6	34

5,7	62
-----	----

5 Click **Remesh** and **OK**.

### Mesh Case 2—Triangular mesh

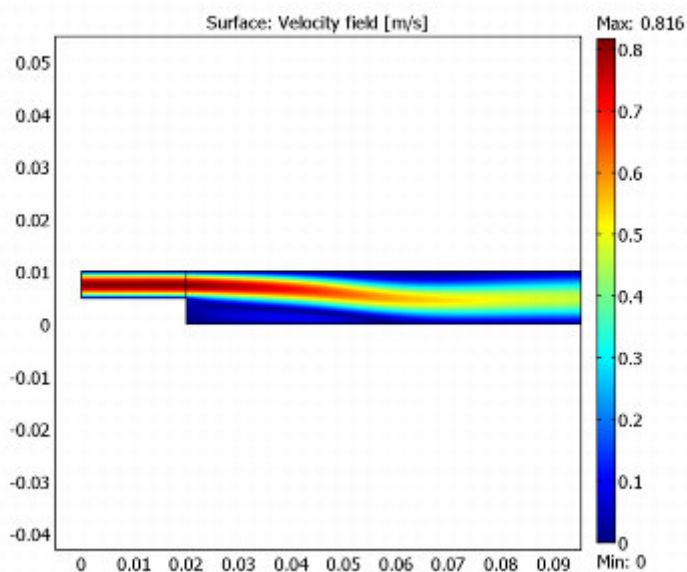
- 1 Go to the **Mesh** menu and choose **Free Mesh Parameters**.
- 2 Enter  $9e-4$  in the edit field for **Maximum element size** on the **Global** page.
- 3 Click **Remesh** and **OK**.

## COMPUTING THE SOLUTION

To start the simulation click the **Solve** button.

## POSTPROCESSING AND VISUALIZATION

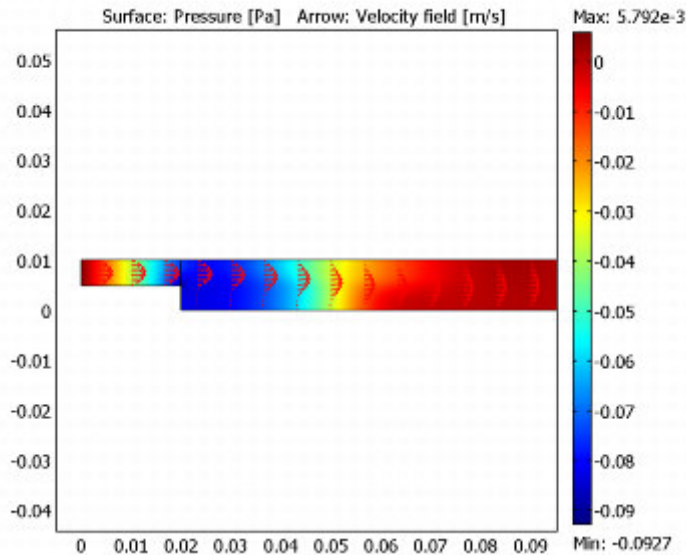
The default visualization plots the magnitude of the velocity field.



To see the velocity field and the pressure field simultaneously, use a combination of arrow and surface plots:

- 1 Go to the **Postprocessing** menu and choose **Plot Parameters** to open the **Plot Parameters** dialog box.
- 2 Click the **Surface** page.
- 3 Under **Surface data** select **Pressure** in the **Predefined quantities** list.
- 4 Click the **Arrow** tab.
- 5 Select the **Arrow plot** check box.
- 6 Click **OK** to view the combined plot.





To visualize the recirculation region, use a conditional expression testing for a negative  $x$ -component in the velocity.

- 1 Open the **Plot Parameters** dialog box again. Add a streamline plot and remove the arrows by selecting the **Streamlines** check box and clearing the **Arrow** check box.
- 2 Click the **Surface** tab.
- 3 In the **Surface data** area, type  $(u < \text{eps}) * (x - 0.02) / 0.0049$  in the **Expression** edit field. Clear the **Smooth** check box (the given expression is deliberately discontinuous).
- 4 Click the **Streamline** tab.
- 5 On the **Streamline Points** tab, click the **Specify start point coordinates** button.
- 6 Type `linspace(0.03, 0.03, 10)` in the **x** edit field and `linspace(0, 0.0101, 10)` in the **y** edit field. These values result in coordinates for 10 equally spaced starting points.
- 7 Click the **Line Color** tab.
- 8 Click the **Color** button and select black as the streamline color.
- 9 Click **OK** to close the **Plot Parameters** dialog box and display the new plot.

## References

1. Gresho, P.M, and Sani, R.L., *Incompressible Flow and the Finite Element Method*, Volume 1 & 2, John Wiley & Sons, New York, 2000.
2. Pironneau, O., *Finite Element Methods for Fluids*, John Wiley & Sons, 1989.
3. Rose, Alan, and Simpson, Ben: *Laminar, Constant-Temperature Flow Over a Backward Facing Step*, 1<sup>st</sup> NAFEMS Workbook of CFD Examples. Glasgow, UK, 2000.
4. J. M. Coulson and J. F. Richardson, *Chemical Engineering*, Volume 2: "Particle Technology and Separation Processes," Butterworth-Heinemann