Created in COMSOL Multiphysics 5.6

3D Supersonic Flow in a Channel with a Bump

Introduction

This example models 3D supersonic flow, including the effect of shocks, in a straight channel with a small obstacle on one of the walls. As the flow hits the obstacle, shock waves are diffracted from the obstacle and the channel walls. The propagating shock waves form a pattern in the velocity profile and density distribution. The model makes use of the adaptive mesh refinement feature in COMSOL Multiphysics. This feature is important as the shock waves should be well resolved by the mesh, but predetermination of their position is very difficult. This example is based on a 2D case that has been widely used in earlier studies of inviscid compressible flow (Ref. 1).

Model Definition

The flow is compressible and inviscid, and the problem is governed by the compressible Euler equations without external forces or heat sources:

$$\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho\mathbf{u}) = 0$$

$$\frac{\partial(\rho\mathbf{u})}{\partial t} + \nabla\cdot(\rho\mathbf{u}\mathbf{u}^T) + \nabla p = 0$$

$$\frac{\partial(\rho E_0)}{\partial t} + \nabla\cdot((\rho E_0 + p)\mathbf{u}) = 0$$

where ρ is the density, $\mathbf{u}$ the velocity vector, p the pressure, and ρE_0 the total energy per unit volume. The perfect gas assumption is used to close the system of 5 variables:

$$p = (\gamma - 1)\left(\rho E_0 - \frac{1}{2}\rho|\mathbf{u}|^2\right)$$

where γ is the ratio of specific heats. The speed of sound, denoted c, is calculated as

$$c = \sqrt{\gamma\frac{p}{\rho}}$$

The Mach number, denoted M, is defined as

$$M = \frac{|\mathbf{u}|}{c}$$

The flow is subsonic if the velocity is below the speed of sound, $M < 1$, and supersonic if it exceeds it, $M > 1$. The inlet Mach number in this case is set to 1.4, which implies that the flow is supersonic and shock waves will appear when the flow hits the obstacle.

Figure 1 shows the geometry of the channel. The length of the channel is 5 m, the height 2 m, and it has a thickness of 0.5 m. The bump is a circular arc with a chord of 1 m and a thickness of 4.2% of the chord, which forms an angle of $30°$ with respect to the inlet surface.

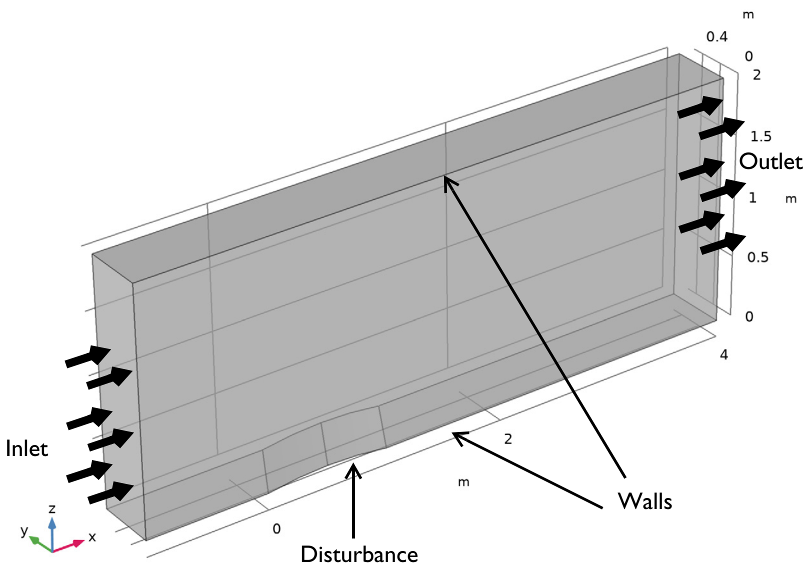


Figure 1: Geometry of the channel with a bump.

This example is solved using the High Mach Number Flow, Laminar interface in COMSOL Multiphysics. This interface solves the compressible Navier-Stokes equations, but the compressible Euler equations can be closely approximated by setting the dynamic viscosity and thermal conductivity to very small values. The adaptive mesh refinement feature is used to refine the mesh near the shocks.

BOUNDARY CONDITIONS

Inlet Condition

The flow at the inlet is supersonic with a flow speed corresponding to a Mach number of 1.4. The inlet flow condition is specified in terms of static properties, where the static pressure is defined as 1 atm and the static temperature is 273.15 K.

Outlet Condition

The flow at the outlet is supersonic, and no constraints are imposed. The outlet condition is modeled using an Outlet node with the Flow condition set to Supersonic.

Walls and Symmetry Conditions

The flow is inviscid and the walls must be modeled as slip walls:

$$\mathbf{u} \cdot \mathbf{n} = \mathbf{0}$$

A symmetry condition is imposed at either side of the channel.

Results and Discussion

The pressure and Mach number in the studied domain are depicted in Figure 2 and Figure 3. The shock formation starts at the position where the flow hits the obstacle. The leading-edge shock wave bounces off the top wall and collides with the trailing-edge shock wave. This qualitative and quantitative description of the diffraction of the shock waves is in very good agreement with the results published in literature regarding the 2D version of this specific case (Ref. 1), but with additional 3D effects.

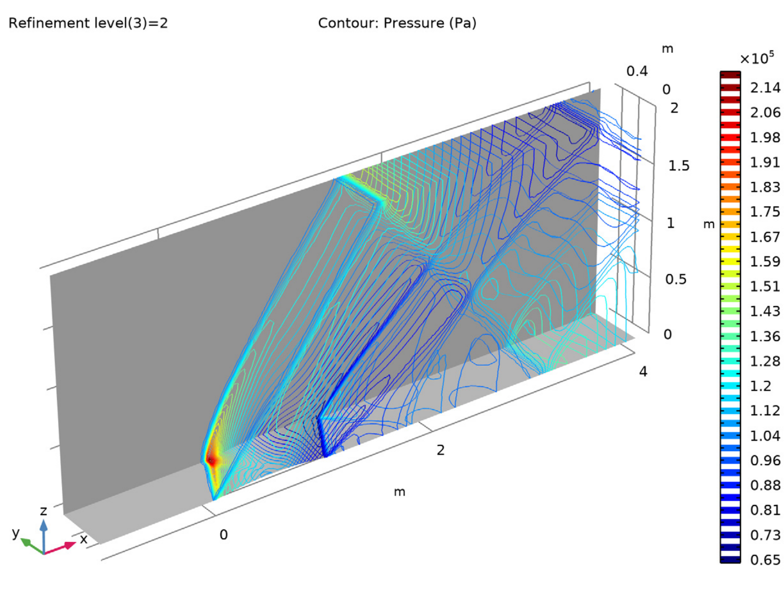


Figure 2: Contours of pressure showing the position of the shocks.

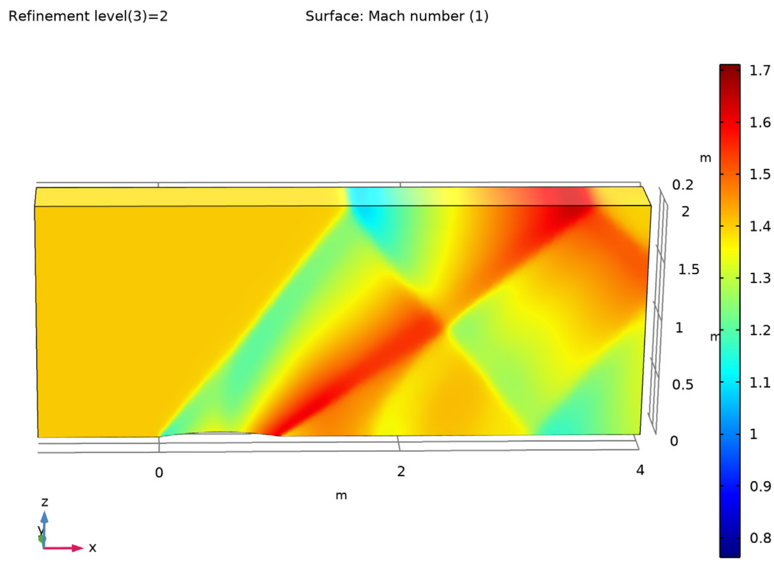


Figure 3: Diffraction pattern in the Mach number arising from supersonic flow hitting a small obstacle in its path.

Another interesting plot, Figure 4, is the mesh obtained after refinement. It is clear that the adaptive mesh feature is able to resolve the zones of the domain where the gradients in Mach number are large. Moreover, the regions where flow variables are uniform are coarsened.

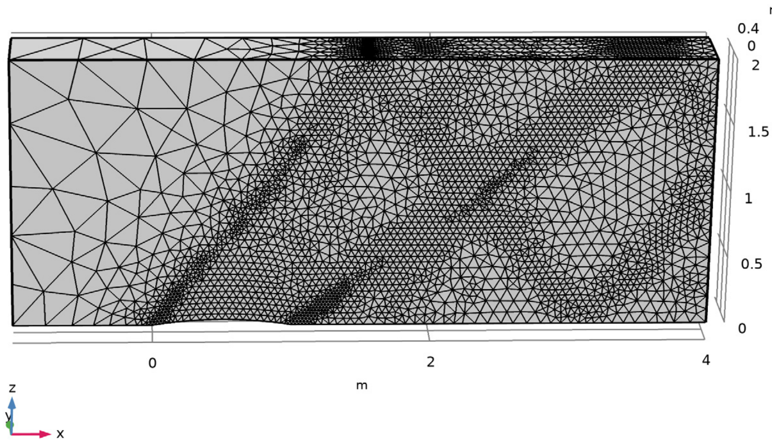


Figure 4: Adapted mesh. The mesh resolves the shock waves more finely than the rest of the modeling domain.

Notes About the COMSOL Implementation

The adaptive mesh refinement feature is used to refine the mesh near the shocks. The L_2 norm is set to compute the error estimate. The **Maximum number of adaptations** is set to 2 with an **Element count growth factor** of 1.7 (the number of elements increases by about 70% at every refinement). **Allow coarsening** is used to coarse the mesh in regions where the flow variables are uniform. Depending on the amount of computer memory available, the maximum number of refinements could be decreased, or increased to improve the resolution of the shock waves.

The linear system of equations for the initial mesh can be solved with a direct solver. However, when the mesh is refined, the number of degrees of freedom of the problem increases and the problem becomes more expensive to solve. An iterative solver is a good option to reduce the computational resources needed to solve the problem.

Reference

1. J.F. Lynn, *Multigrid Solution of the Euler Equations with Local Preconditioning*, Dissertation, University of Michigan, 1995.

Application Library path: `CFD_Module/High_Mach_Number_Flow/euler_bump_3d`

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 **3D**.

2 In the **Select Physics** tree, select **Fluid Flow>High Mach Number Flow> High Mach Number Flow, Laminar (hmnf)**.

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
Min	1.4	1.4	Mach number, inlet
pin	1[atm]	1.0133E5 Pa	Static pressure, inlet
Tin	273.15[K]	273.15 K	Static temperature, inlet

Name	Expression	Value	Description
Rs	287[J/kg/K]	287 J/(kg·K)	Specific gas constant
gamma	1.4	1.4	Ratio of specific heats
uin	Min*sqrt(gamma*Rs*Tin)	463.8 m/s	Velocity, inlet
alpha	30[deg]	0.5236 rad	Angle of the obstacle

GEOMETRY 1

First, build the rectangular channel.

Block 1 (blk1)

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

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

3 In the **Width** text field, type `5`.

4 In the **Depth** text field, type `0.5`.

5 In the **Height** text field, type `2`.

6 Locate the **Position** section. In the **x** text field, type `-1`.

Now, build a cylinder that will be used to create the obstacle.

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **xz-plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

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

3 In the **Radius** text field, type `(0.5^2/0.042+0.042)/2`.

4 Locate the **Position** section. In the **xw** text field, type `0.5`.

5 In the **yw** text field, type `0.042-(0.5^2/0.042+0.042)/2`.

6 In the **Work Plane** toolbar, click **Build All**.

Extrude 1 (ext1)

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

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
-0.5

4 Click to expand the **Displacements** section. In the table, enter the following settings:

Displacements xw (m)	Displacements yw (m)
0.5*tan(alpha)	0

5 Click **Build Selected**.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **blk1** only (rectangular channel) in the **Objects to add** subsection.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.

5 Select the object **ext1** only (the cylinder)

6 Click **Build All Objects**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar (see the figure below).

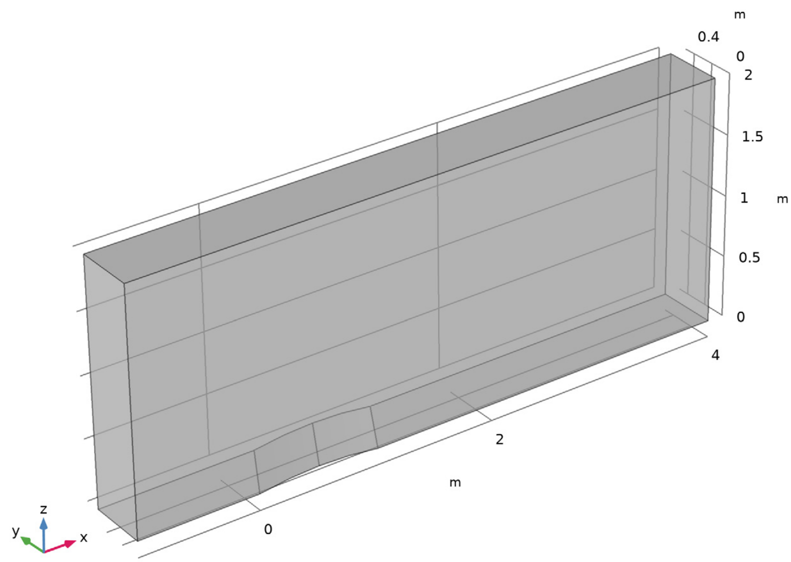
DEFINITIONS

View 3

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **View**.

Camera

Define a preferred view. The model MPH-file available in the Application Libraries contains different views used to obtain the figures shown in this documentation.



HIGH MACH NUMBER FLOW, LAMINAR (HMNF)

This problem does not require **Pseudo-Time stepping for stationary equation form**. The convergence is faster if this option is deselected.

1. Click the **Show More Options** button in the **Model Builder** toolbar.
2. In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.
3. Click **OK**.
4. In the **Model Builder** window, expand the **View 3** node, then click **Component 1 (comp1)> High Mach Number Flow, Laminar (hmnf)**.
5. In the **Settings** window for **High Mach Number Flow, Laminar**, click to expand the **Advanced Settings** section.
6. Clear the **Use pseudo time stepping for stationary equation form** check box.

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>High Mach Number Flow, Laminar (hmnf)** click **Fluid 1**.

2 In the **Settings** window for **Fluid**, locate the **Heat Conduction** section.

3 From the k list, choose **User defined**. In the associated text field, type `1e-8`.

4 Locate the **Thermodynamics** section. From the R_s list, choose **User defined**. In the associated text field, type `Rs`.

5 From the **Specify Cp or** γ list, choose **Ratio of specific heats**.

6 From the γ list, choose **User defined**. In the associated text field, type `gamma`.

7 Locate the **Dynamic Viscosity** section. From the μ list, choose **User defined**. In the associated text field, type `1e-8`.

The dynamic viscosity and thermal conductivity are set to small values to mimic an inviscid and nonconducting fluid.

Wall 1

1 In the **Model Builder** window, click **Wall 1**.

2 In the **Settings** window for **Wall**, locate the **Boundary Condition** section.

3 From the **Wall condition** list, choose **Slip**.

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

uin	x
0	y
0	z

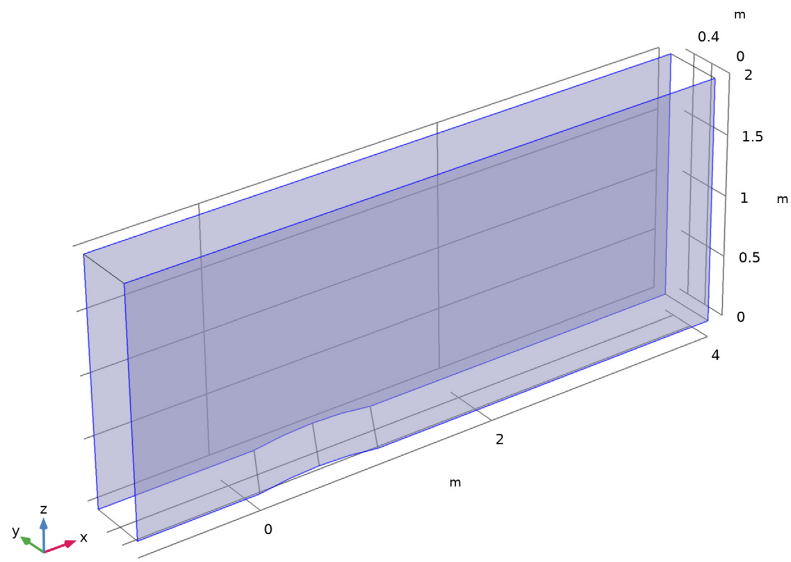
4 In the p text field, type `pin`.

5 In the T text field, type `Tin`.

Symmetry 1

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

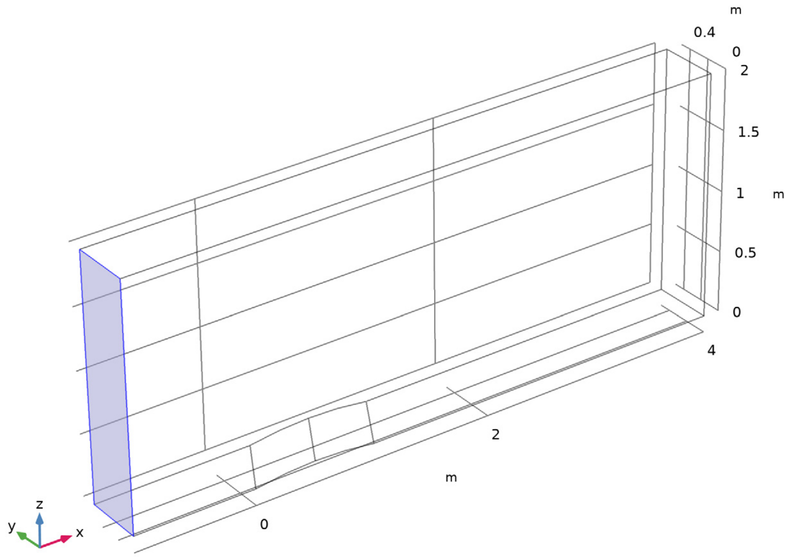
2 Select Boundaries 2 and 5 only.



Inlet 1

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

2 Select Boundary 1 only.



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

4 From the **Flow condition** list, choose **Supersonic**.

5 Locate the **Flow Properties** section. In the $p_{0,\mathrm{stat}}$ text field, type `pin`.

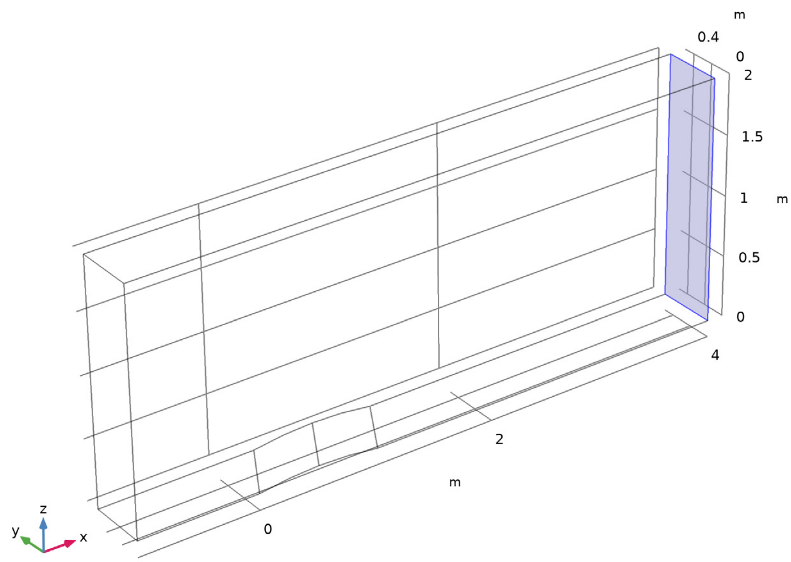
6 In the $T_{0,\mathrm{stat}}$ text field, type `Tin`.

7 In the Ma_0 text field, type `Min`.

Outlet 1

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

2 Select Boundary 9 only.



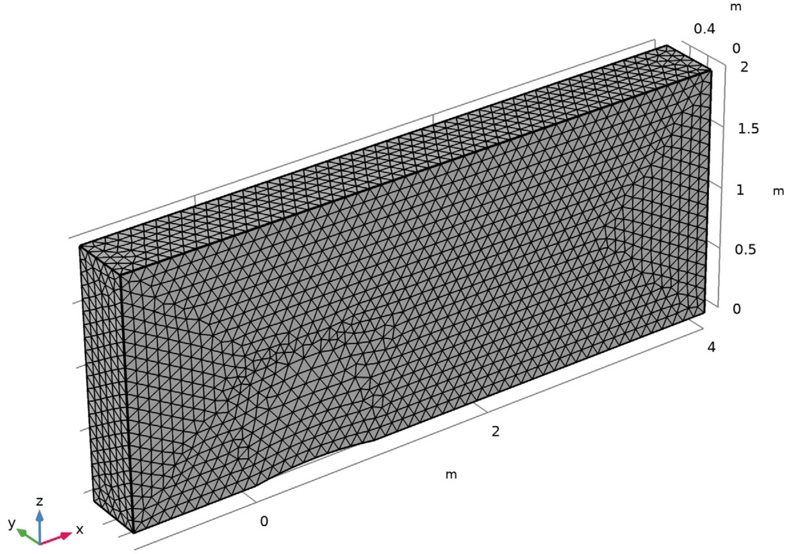
3 In the **Settings** window for **Outlet**, locate the **Flow Condition** section.

4 From the **Flow condition** list, choose **Supersonic**.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

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



STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Adaptation and Error Estimates** section.

3 From the **Adaptation and error estimates** list, choose **Adaptation and error estimates**.

4 Find the **Mesh adaptation** subsection. Select the **Maximum number of adaptations** check box.

5 In the associated text field, type 2.

Depending on the amount of computer memory available the **Maximum number of adaptations** may be increased to improve the resolution of shock waves.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

The amount of computational resources needed to solve the problem after refining the mesh can be reduced by means of an iterative solver:

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, locate the **General** section.

5 From the **Linear solver** list, choose **AMG, fluid flow variables (hmnf)**.

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

RESULTS

Pressure (hmnf)

1 In the **Model Builder** window, under **Results** click **Pressure (hmnf)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/Adaptive Mesh Refinement Solutions 1 (sol2)**.

Surface

1 In the **Model Builder** window, expand the **Pressure (hmnf)** node.

2 Right-click **Surface** and choose **Delete**.

Pressure (hmnf)

1 In the **Model Builder** window, click **Pressure (hmnf)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 Clear the **Plot dataset edges** check box.

Surface 1

1 Right-click **Pressure (hmnf)** and choose **Surface**.

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

3 In the **Expression** text field, type 1.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

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

6 From the **Color** list, choose **Gray**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

2 Select Boundaries 3 and 5–8 only.

3 In the **Pressure (hmnf)** toolbar, click **Plot** (see Figure 2).

Slice

1 In the **Model Builder** window, expand the **Mach Number (hmnf)** node.

2 Right-click **Slice** and choose **Delete**.

Surface 1

1 In the **Model Builder** window, right-click **Mach Number (hmnf)** and choose **Surface**.

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

3 In the **Expression** text field, type `hmnf.Ma`.

4 In the **Mach Number (hmnf)** toolbar, click **Plot** (see Figure 3).