Created in COMSOL Multiphysics 6.1

Transonic Flow in a Sajben Diffuser

Introduction

In the present example the high speed turbulent gas flow in a converging and diverging nozzle is modeled using the High Mach Number Flow interface. The diffuser is transonic in the sense that the flow at the inlet is subsonic, but due to the contraction and the low outlet pressure, the flow accelerates and becomes sonic ($\mathrm{Ma} = 1$) in the throat of the nozzle. After a short region of supersonic flow, a normal shock wave brings the flow back to subsonic flow. This setup has been studied in a number of experiments and numerical simulations by M. Sajben and co-workers (see for example Ref. 1, Ref. 2, Ref. 3, and Ref. 4), in an effort to study unsteady fluctuations in supersonic inlets with applications in supersonic aircraft propulsion systems. The geometry and setup is often referred to as a Sajben diffuser and constitutes a common test case for the simulation of high Mach number internal flows. In this example, the time-averaged transonic flow through a Sajben diffuser is solved for using two different exit pressures. The flow in the diffuser is fully turbulent with a inlet Reynolds number of 7×10^5 based on the inlet fluid properties and the channel height. The model uses the Spalart-Allmaras turbulence model to compute the turbulent viscosity. For the first outlet pressure value a normal shock is present, but the flow remains attached throughout the diverging part. For the second, lower, outlet pressure value, the shock is strong enough to cause a shock-induced separation in the diverging part. Based on the ability to induce flow separation, the shock in the first case is referred to as weak, while in the second case it is termed strong following the definition in Ref. 2.

Model Definition

Figure 1 shows the physical geometry of the converging and diverging nozzle model. The nozzle dimensions correspond to those used in the experiments in Ref. 2 and in the benchmarks simulation of Ref. 5 and Ref. 6. In the central contraction part, the minimum vertical height separating the lower and upper walls, the throat height h_{th}, is 1.7322 in (44 mm). The channel height at the inlet is $1.4h_{\mathrm{th}}$, and the outlet height is $1.5h_{\mathrm{th}}$.

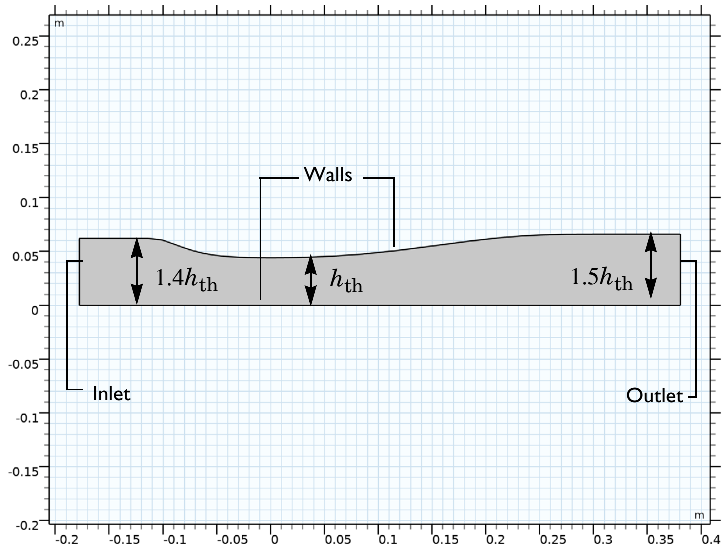


Figure 1: Geometry and dimensions of the Sajben diffuser model.

FLUID PROPERTIES

The fluid occupying the channel is assumed to be air, by specifying a specific gas constant of 287 J/(kg·K) and a ratio of specific heats of 1.4. The dynamic viscosity is computed using Sutherland's Law:

$$\mu = \mu_{\mathrm{ref}}\left(\frac{T}{T_{\mathrm{ref}}}\right)^{\frac{3}{2}}\frac{T_{\mathrm{ref}}+S_{\mu}}{T+S_{\mu}}$$

where the $T_{\mathrm{ref}} = 500$ R (about 278 K) corresponds to the inlet total temperature, and the Sutherland constant is $S_{\mu} = 111$ K. The reference viscosity μ_{ref} is defined from the inlet Reynolds number:

$$\mathrm{Re}_{\mathrm{in}} = \frac{\rho_{\mathrm{in}}U_{\mathrm{in}}h_{\mathrm{in}}}{\mu_{\mathrm{ref}}}$$

which in turn is used in a parametric sweep, where the model is solved for increasing Reynolds number. The final inlet Reynolds number to be computed is 7×10^5, corresponding to the one used in the simulation in Ref. 5. The thermal conductivity of the gas is defined using the definition of the Prandtl number

$$\mathrm{Pr} = \frac{C_p \mu}{k}$$

which is assumed to be 0.71 in this case. This is a typical number for air around 293 K.

BOUNDARY CONDITIONS

Inlet Condition

The flow at the inlet is subsonic with a flow speed corresponding to a Mach number of 0.46. The inlet conditions are specified in terms of total properties, where the total pressure is defined as 19.58 psi and the total temperature is 500 R. The inlet conditions are applied using an Inlet feature, where the Flow condition is specified to be Characteristics based. This provides a numerically consistent boundary condition by evaluating the flow characteristics at the inlet (for more background on this boundary condition, see the *CFD Module User's Guide*).

Outlet Condition

At the outlet the static pressure is specified. The model is solved for the two outlet pressure values specified in Table 1 below.

TABLE 1: OUTLET PRESSURE.

PRESSURE	FRACTION OF INLET TOTAL PRESSURE	DESCRIPTION
16.05 psi	0.82	Case 1: weak shock
14.10 psi	0.72	Case 2: strong shock

These outlet pressure values are known from experiments and simulations to be low enough to produce sonic conditions at the throat of the nozzle. However, they are not low enough for the flow to stay supersonic throughout the diverging part. The supersonic flow in the divergent part is terminated by a normal shock wave, so that the flow in the following part including the outlet becomes subsonic. The pressure is specified in the model using an Outlet node with the Flow condition set to Subsonic.

Results and Discussion

Below, some of the results from the transonic diffuser model computed in COMSOL Multiphysics are shown and discussed. The results are compared to experimental data from Ref. 2 (strong shock case) and Ref. 4 (weak shock case). The experimental data was extracted as tabulated data from Ref. 5 and Ref. 6 and plotted in COMSOL using interpolation functions.

Figure 2 shows the Mach numbers and velocity streamlines resulting from applying the first outlet pressure, 16.05 psi. It can be seen that the flow accelerates in the converging part, reaches sonic conditions at the throat, after which a region of supersonic flow follows in the diverging part. The supersonic region is terminated by a normal shock wave, which brings the flow back to subsonic conditions. In the remaining part of the channel the flow decelerates subsonically toward the outlet. The zero contour of the x-component velocity is also plotted in the figure, but this is only present on the walls and not visible inside the domain. Hence no separation zone is present, and the flow remains attached throughout the divergent part of the channel. The shock is not able to cause flow separation and is therefore termed weak. These results correlate well with those in Ref. 2 and Ref. 5.

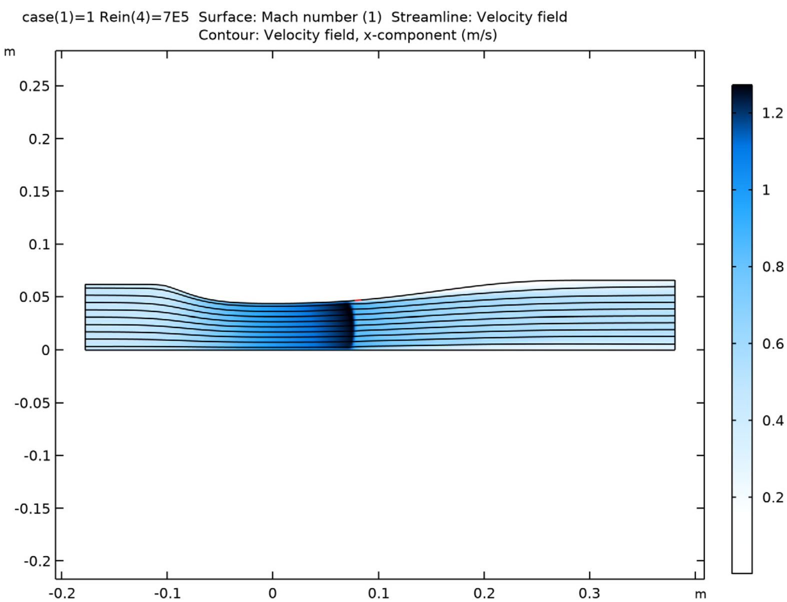


Figure 2: Mach number, flow streamlines, and zero x-component velocity contour resulting from the weak shock case.

Figure 3 shows the same quantities as Figure 2, but uses the results from the second outlet pressure case, p_{out} = 14.10 psi. Due to the lower outlet pressure, the normal shock wave is positioned further downstream in the divergent channel part. More importantly, a flow separation zone can be seen behind the shock, as indicated by the zero x-component velocity contour. The shock wave in this case is apparently strong enough to induce flow separation. This result is in accordance with those presented in Ref. 2 and Ref. 6.

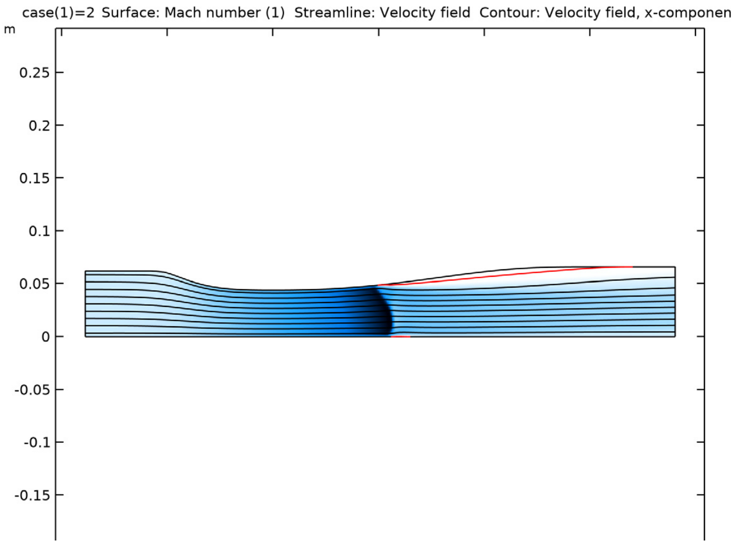


Figure 3: Mach number, flow streamlines, and zero x-component velocity contour (in red) resulting from the strong shock case.

Figure 4 shows the development of the static pressure on the upper wall normalized by the inlet total pressure. Results from both the weak and strong cases are plotted and compared with the experimental data of Ref. 1 and Ref. 2. The results from both outlet cases are in general in very good agreement with the experimental results in the diffuser. Note however that the shock positions in the model are slightly shifted in the downstream direction in comparison with the experiments.

For analysis of the results in the interior of the channel, Figure 5 plots the streamwise velocity profiles from the strong shock case at two different positions in the divergent part of the channel together with experimental results. The velocity profile at the first position, $x = 4.611h_{\mathrm{th}}$, compares very well with the experimental results. Both the velocity magnitude and the size of the separation zone, including reversed flow, are accurately reproduced. Further downstream, at the $x = 6.340h_{\mathrm{th}}$ position, the velocity magnitude in the central part of the channel is also in good agreement with the experimental results. Closer to the upper wall, the model results include flow reversal at this position. This is not found in the experimental result, indicating that the separation zone in the model extends further downstream than that in the experiment.

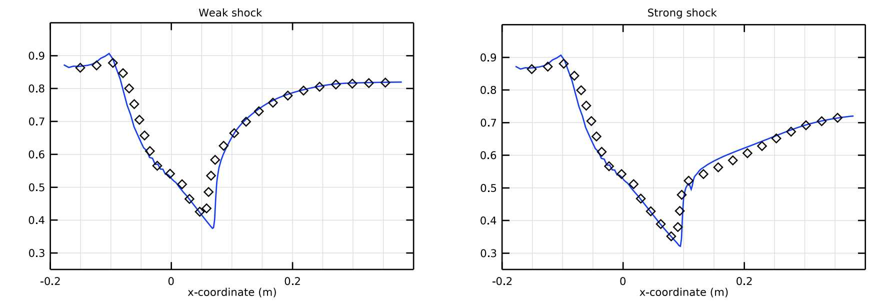


Figure 4: Top wall static pressure normalized by the inlet total pressure. Model results (lines) and experimental results (diamonds) are shown.

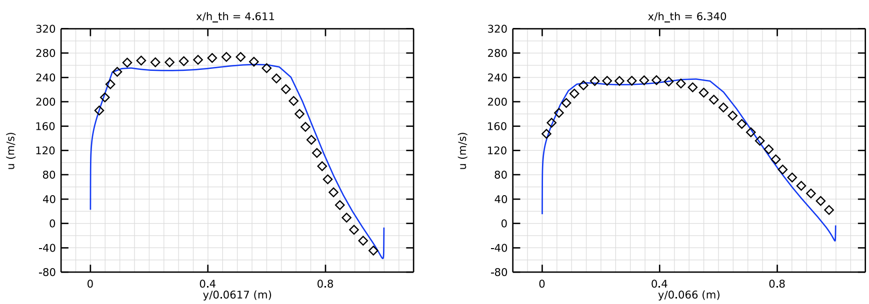


Figure 5: Mean x-component velocity at two positions downstream of the strong shock. Model results (lines) and experimental results (diamonds) are shown.

Notes About the COMSOL Implementation

The present model is highly nonlinear and sensitive to the solution procedure. The sensitivity is accentuated by the fact that the model seeks to determine the equilibrium position of a normal shock wave, positioned in a channel with smoothly varying channel height.

You solve the model in two steps. First, apply the higher outlet pressure to simulate the weak shock case. To solve this case, use a parametric sweep where the inlet Reynolds number increases stepwise by decreasing the dynamic viscosity. When you have obtained a converged result for the highest Reynolds number, use this solution as the initial condition for the second case with the lower outlet pressure value. In both cases, use pseudo-time stepping with manually defined CFL number expressions to compute stationary solutions.

References

1. M. Sajben, J.C. Kroutil, and C.P. Chen, “A High-Speed Schlieren Investigation of Diffuser Flows with Dynamic Distortion”, *AIAA Paper 77-875*, 1977.

2. T.J. Bogar, M. Sajben, and J.C. Kroutil, “Characteristic Frequencies of Transonic Diffuser Flow Oscillations,” *AIAA Journal*, vol. 21, no. 9, pp. 1232–1240, 1983.

3. J.T. Salmon, T.J. Bogar, and M. Sajben, “Laser Doppler Velocimetry in Unsteady, Separated, Transonic Flow,” *AIAA Journal*, vol. 21, no. 12, pp. 1690–1697, 1983.

4. T. Hsieh, A.B. Wardlaw Jr., T.J. Bogar, P. Collins, and T. Coakley, “Numerical Investigation of Unsteady Inlet Flowfields,” *AIAA Journal*, vol. 25, no. 1, pp. 75–81, 1987.

5. NPARC Alliance Validation Archive, “Sajben Transonic Diffuser: Study #1,”, 2008, http://www.grc.nasa.gov/WWW/wind/valid/transdif/transdif01/transdif01.html

6. NPARC Alliance Validation Archive, “Sajben Transonic Diffuser: Study #2,”, 2008, http://www.grc.nasa.gov/WWW/wind/valid/transdif/transdif02/transdif02.html

Application Library path: `CFD_Module/High_Mach_Number_Flow/sajben_diffuser`

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>High Mach Number Flow>Turbulent Flow>High Mach Number Flow, Spalart-Allmaras (hmnf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary with Initialization**.

6 Click **Done**.

GLOBAL DEFINITIONS

Geometry parameters

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
`x0`	`-6.99809[in]`	-0.17775 m	`Inlet x-position`
`xEnd`	`14.98353[in]`	0.38058 m	`Outlet x-position`
`h_in`	`2.44483[in]`	0.062099 m	`Diffuser inlet height`
`h_out`	`2.59830[in]`	0.065997 m	`Diffuser outlet height`
`h_th`	`1.732[in]`	0.043993 m	`Throat height`

4 In the **Label** text field, type `Geometry parameters`.

Fluid flow parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Fluid flow parameters` in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
`Rein`	`7e5`	7E5	`Inlet Reynolds number`
`case`	`1`	1	`Case number; 1 = weak shock, 2 = strong shock`
`Min`	`0.46`	0.46	`Inlet Mach number`
`gamma`	`1.4`	1.4	`Ratio of specific heats`
`Pr`	`0.72`	0.72	`Prandtl number`
`Rs`	`287[J/kg/K]`	287 J/(kg·K)	`Specific gas constant`

Name	Expression	Value	Description
Tin_tot	500[R]	277.78 K	Inlet total temperature
Tin_stat	Tin_tot/(1+0.5*Min^2*(-1+gamma))	266.5 K	Inlet static temperature
pin_tot	19.58[psi]	1.35E5 Pa	Inlet total pressure
pin_stat	pin_tot/(1+0.5*Min^2*(-1+gamma))^(gamma/(-1+gamma))	1.1677E5 Pa	Inlet static pressure
rhoin	pin_stat/Rs/Tin_stat	1.5267 kg/m³	Inlet density
mu_ref	rhoin*u_in*h_in/Rein	2.0387E-5 kg/(m·s)	Reference dynamic viscosity
u_in	Min*sqrt(gamma*Rs*Tin_stat+eps)	150.53 m/s	Inlet velocity
pOut	if(case==1,16.05,0)[psi]+if(case==2,14.1,0)[psi]	1.1066E5 Pa	Outlet pressure

Interpolation 1 (int1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Interpolation**.

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

3 From the **Data source** list, choose **File**.

4 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
top_pos	1

5 Click **Browse**.

6 Browse to the model's Application Libraries folder and double-click the file `sajben_diffuser_upper_wall.txt`.

7 Click **Import**.

8 Locate the **Units** section. In the **Function** table, enter the following settings:

Function	Unit
top_pos	in

Interpolation 2 (int2)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Interpolation**.

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

3 From the **Data source** list, choose **File**.

4 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
ptop_weak	1

5 Click **Browse**.

6 Browse to the model's Application Libraries folder and double-click the file `sajben_diffuser_ptop_weak.txt`.

7 Click **Import**.

Interpolation 3 (int3)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Interpolation**.

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

3 From the **Data source** list, choose **File**.

4 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
ptop_strong	1

5 Click **Browse**.

6 Browse to the model's Application Libraries folder and double-click the file `sajben_diffuser_ptop_strong.txt`.

7 Click **Import**.

Interpolation 4 (int4)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Interpolation**.

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

3 From the **Data source** list, choose **File**.

4 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
u_at4611	1

5 Click **Browse**.

6 Browse to the model's Application Libraries folder and double-click the file `sajben_diffuser_u-xh4611.txt`.

7 Click **Import**.

Interpolation 5 (int5)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Interpolation**.

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

3 From the **Data source** list, choose **File**.

4 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
u_at6340	1

5 Click **Browse**.

6 Browse to the model's Application Libraries folder and double-click the file `sajben_diffuser_u-xh6340.txt`.

7 Click **Import**.

DEFINITIONS

Variables 1

1 In the **Home** toolbar, click a= **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
CFLnum	if(case==1,CFLweak,0)+if(case==2,CFLstrong,0)		CFL number for pseudo time stepping
CFLweak	1.3^min(niterCMP-1,9)+if(niterCMP>25,5*1.2^min(niterCMP-26,12),0)		CFL number, weak case
CFLstrong	1+if(niterCMP>10,1.2^min(niterCMP-10,12),0)+if(niterCMP>120,1.3^min(niterCMP-120,9),0)+if(niterCMP>220,1.3^min(niterCMP-220,9),0)		CFL number, strong case

The manual CFL number expression for the strong shock corresponds to the implemented automatic expression for turbulent flows. In this case the solution already contains a shock that will move due to the change in the outlet pressure, and a cautious increase of the CFL number is needed. In the weak case simulation, a shock is not yet formed, and the simulation time can be reduced by using a more aggressive ramping of the CFL number.

GEOMETRY 1

Parametric Curve 1 (pc1)

1. In the **Geometry** toolbar, click **More Primitives** and choose **Parametric Curve**.
2. In the **Settings** window for **Parametric Curve**, locate the **Parameter** section.
3. In the **Minimum** text field, type `x0[1/in]`.
4. In the **Maximum** text field, type `xEnd[1/in]`.
5. Locate the **Expressions** section. In the **x** text field, type `s[in]`.
6. In the **y** text field, type `top_pos(s)`.
7. Click **Build Selected**.

Polygon 1 (pol1)

1. In the **Geometry** toolbar, click **Polygon**.
2. In the **Settings** window for **Polygon**, locate the **Object Type** section.
3. From the **Type** list, choose **Open curve**.
4. Locate the **Coordinates** section. From the **Data source** list, choose **Vectors**.
5. In the **x** text field, type `x0 x0 x0 xEnd xEnd xEnd`.
6. In the **y** text field, type `h_in 0 0 0 0 h_out`.
7. Click **Build Selected**.

Convert to Solid 1 (csol1)

1. In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.
2. Click in the **Graphics** window and then press Ctrl+A to select both objects.
3. In the **Settings** window for **Convert to Solid**, click **Build Selected**.

Add a rectangular domain in the divergent part of the nozzle. This will be used to increase the resolution in the region where the shock is located.

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 `0.16`.
4. In the **Height** text field, type `0.1`.

5 Locate the **Position** section. In the **x** text field, type `0.025`.

6 Click **Build Selected**.

Partition Objects 1 (par1)

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

2 Select the object **csol1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 Find the **Tool objects** subsection. Click to select the **Activate Selection** toggle button.

5 Select the object **r1** only.

6 Click **Build Selected**.

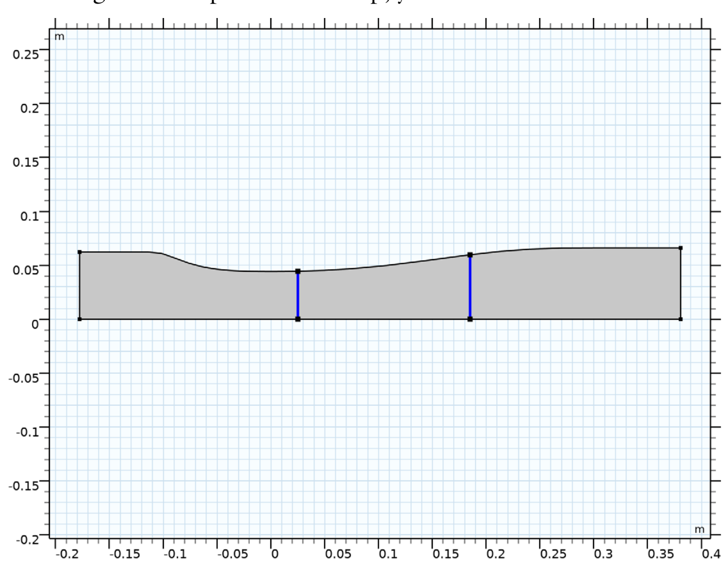
Add a **Mesh Control Edges** feature to specify the interior boundaries as mesh control entities. In this manner these entities can be used to control the mesh, but at the same time they will automatically be omitted when defining the physics and when postprocessing results.

Mesh Control Edges 1 (mce1)

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

2 On the object **fin**, select Boundaries 3 and 5 only.

It might be easier to select the boundaries 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**.

HIGH MACH NUMBER FLOW, SPALART-ALLMARAS (HMNF)

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>High Mach Number Flow, Spalart-Allmaras (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 `hmnf.Cp*hmnf.mu/Pr`.

Here the conductivity is defined using a constant Prandtl number.

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. In the μ_{ref} text field, type `mu_ref`.

8 In the $T_{\mu,ref}$ text field, type `Tin_stat`.

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 **u** vector as

u_in	x
0	y

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

5 In the *nutilde* text field, type `subst(hmnf.nutildeinit,p,pin_stat)`.

This ensures that when evaluating the initial condition for `nutilde`, the pressure used corresponds to `pin_stat`.

6 In the T text field, type `Tin_stat`.

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 Properties** section.

4 From the **Input state** list, choose **Total**.

5 In the $p_{0,\text{tot}}$ text field, type `pin_tot`.

6 In the $T_{0,\text{tot}}$ text field, type `Tin_tot`.

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

Outlet 1

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

2 Select Boundary 3 only.

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

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

5 Locate the **Flow Properties** section. From the **Boundary condition** list, choose **Pressure**.

6 In the p_0 text field, type `pOut`.

CFL NUMBER

To apply the manually defined CFL number, first enable the **Advanced Physics Options**.

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, click **High Mach Number Flow, Spalart-Allmaras (hmnf)**.

5 In the **Settings** window for **High Mach Number Flow, Spalart-Allmaras**, click to expand the **Advanced Settings** section.

6 From the **CFL number expression** list, choose **Manual**.

7 In the CFL_{loc} text field, type `CFLnum`.

MESH 1

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Distribution 1

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

2 Select Boundaries 4 and 6 only.

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

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

5 In the **Number of elements** text field, type `40`.

6 In the **Element ratio** text field, type `1/4`.

Distribution 2

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

2 Select Boundaries 5 and 7 only.

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

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

5 In the **Number of elements** text field, type `90`.

Distribution 3

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

2 Select Boundaries 2 and 8 only.

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

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

5 In the **Number of elements** text field, type `50`.

6 In the **Element ratio** text field, type `3`.

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

Distribution 4

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

2 Select Boundaries 1 and 3 only.

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

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

5 In the **Number of elements** text field, type `25`.

6 In the **Element ratio** text field, type `2.5`.

7 Select the **Symmetric distribution** check box.

8 Click **Build All**.

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

Boundary Layers 1

1 In the **Mesh** toolbar, click **Boundary Layers**.

In this case the mesh transition region, between the boundary layer and the interior mesh, is explicitly controlled by the specified distributions. The default mesh smoothing of the transition region can hence be disabled.

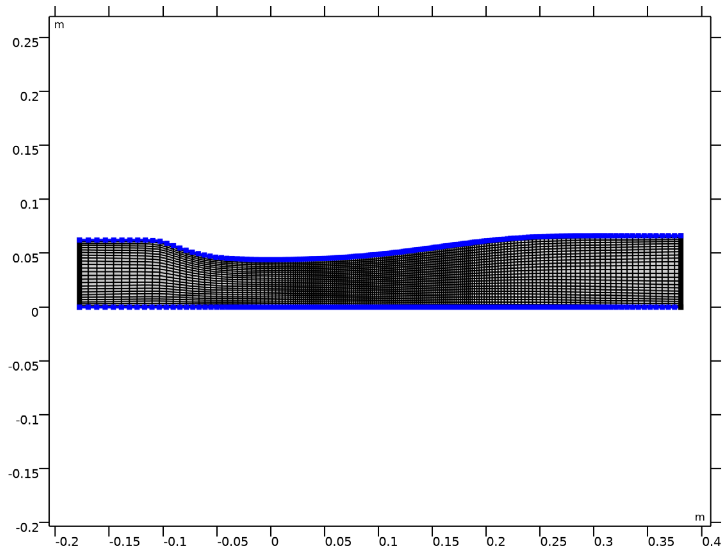
2 In the **Settings** window for **Boundary Layers**, click to expand the **Transition** section.

3 Clear the **Smooth transition to interior mesh** check box.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 Select Boundaries 2, 4, 6, and 8–10 only.



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

4 In the **Number of layers** text field, type `20`.

5 In the **Thickness adjustment factor** text field, type `0.11`.

6 Click **Build All**.

STUDY 1

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
case (Case number; 1 = weak shock, 2 = strong shock)	1	

Step 2: Stationary

Set up an auxiliary continuation sweep for the 'Rein' parameter.

1 In the **Model Builder** window, click **Step 2: Stationary**.

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

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Rein (Inlet Reynolds number)	5e3 5e4 2e5 7e5	

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

RESULTS

To reproduce the plot in Figure 2 perform the steps below.

Surface 1

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

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Aurora>JupiterAuroraBorealis** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

7 From the **Color table transformation** list, choose **Reverse**.

Streamline 1

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

2 Select Boundary 1 only.

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

4 In the **Number** text field, type 9.

Contour 1

1 Right-click **Mach Number (hmnf)** and choose **Contour**.

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

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

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

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

6 Clear the **Color legend** check box.

7 In the **Mach Number (hmnf)** toolbar, click **Plot**.

Select all **2D Plot Groups**, right-click and select **Group**. This will group all the plots that belong to the weak shock case together.

Weak Shock

1 In the **Model Builder** window, under **Results** click **Group 1**.

2 In the **Settings** window for **Group**, type `Weak Shock` in the **Label** text field.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary with Initialization**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
case (Case number; 1 = weak shock, 2 = strong shock)	2	

Before computing the solution for the strong shock case, apply the last solution from the weak shock case as initial value.

Step 1: Wall Distance Initialization

1 In the **Model Builder** window, click **Step 1: Wall Distance Initialization**.

2 In the **Settings** window for **Wall Distance Initialization**, click to expand the **Values of Dependent Variables** section.

3 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

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

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

6 From the **Parameter value (Rein)** list, choose **Last**.

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

RESULTS

To reproduce the plot in Figure 3 perform the steps below.

Surface 1

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

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Aurora>JupiterAuroraBorealis** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

7 From the **Color table transformation** list, choose **Reverse**.

Streamline 1

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

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>High Mach Number Flow, Spalart-Allmaras (Fluid Flow)>Velocity and pressure>u,v - Velocity field**.

3 Select Boundary 1 only.

4 Locate the **Streamline Positioning** section. In the **Number** text field, type `9`.

Contour 1

1 Right-click **Mach Number (hmnf) 1** 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)>High Mach Number Flow, Spalart-Allmaras (Fluid Flow)>Velocity and pressure>Velocity field - m/s>u - Velocity field, x-component**.

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

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

5 Clear the **Color legend** check box.

6 In the **Mach Number (hmnf) 1** toolbar, click **Plot**.

Select all the new 2D plots, right-click and select **Group**.

Strong Shock

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

2 In the **Settings** window for **Group**, type `Strong Shock` in the **Label** text field.

Create cut line datasets to plot results at two downstream positions in the diverging part of the nozzle.

Cut Line 2D 1

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets** and choose **Cut Line 2D**.

3 In the **Settings** window for **Cut Line 2D**, locate the **Data** section.

4 From the **Dataset** list, choose **Study 2/Solution 5 (sol5)**.

5 Locate the **Line Data** section. In row **Point 1**, set **x** to `4.611*h_th`.

6 In row **Point 2**, set **x** to `4.611*h_th` and **y** to `2*h_th`.

7 Click **Plot**.

The position of the cut line is indicated with a red line.

Cut Line 2D 2

1 In the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 5 (sol5)**.

4 Locate the **Line Data** section. In row **Point 1**, set **x** to `6.340*h_th`.

5 In row **Point 2**, set **x** to `6.340*h_th` and **y** to `2*h_th`.

6 Click **Plot**.

The following steps reproduce the normalized static pressure plots in Figure 4.

1D Plot Group 13

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Parameter selection (Rein)** list, choose **Last**.

Line Graph 1

1 Right-click **1D Plot Group 13** and choose **Line Graph**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `p/pin_tot`.

5 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Geometry>Coordinate>x - x-coordinate**.

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `ptop_weak(x/h_th)`.

4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 From the **Color** list, choose **Black**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Diamond**.

7 From the **Positioning** list, choose **Interpolated**.

8 In the **Number** text field, type `30`.

1D Plot Group 13

1 In the **Model Builder** window, click **1D Plot Group 13**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Weak shock`.

5 Locate the **Axis** section. Select the **Manual axis limits** check box.

6 In the **x minimum** text field, type `-0.2`.

7 In the **x maximum** text field, type `0.4`.

8 In the **y minimum** text field, type `0.25`.

9 In the **y maximum** text field, type `1`.

10 Locate the **Grid** section. Select the **Manual spacing** check box.

11 In the **x spacing** text field, type `0.05`.

12 In the **y spacing** text field, type `0.1`.

13 In the **1D Plot Group 13** toolbar, click **Plot**.

Compare the result with that in the left panel of Figure 4.

To reproduce the plot in the right panel, use the plot you just created as the starting point.

1D Plot Group 14

1 Right-click **Results>1D Plot Group 13** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 5 (sol5)**.

4 Locate the **Title** section. In the **Title** text area, type `Strong shock`.

Line Graph 2

1 In the **Model Builder** window, expand the **1D Plot Group 14** node, then click **Line Graph 2**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `ptop_strong(x/h_th)`.

4 In the **1D Plot Group 14** toolbar, click **Plot**.

1D Plot Group 14

1 In the **Model Builder** window, click **1D Plot Group 14**.

2 Click **Plot**.

Compare with the right panel of Figure 4.

Similarly, reproduce the x-velocity plots in Figure 5.

1D Plot Group 15

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Line 2D 1**.

Line Graph 1

1 Right-click **1D Plot Group 15** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> High Mach Number Flow, Spalart-Allmaras (Fluid Flow)>Velocity and pressure> Velocity field - m/s>u - Velocity field, x-component**.

3 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type `y/0.0617`.

Line Graph 2

1 In the **Model Builder** window, right-click **1D Plot Group 15** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `u_at4611(y/0.0617)`.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `y/0.0617`.

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

7 From the **Color** list, choose **Black**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Diamond**.

9 From the **Positioning** list, choose **Interpolated**.

10 In the **Number** text field, type `30`.

1D Plot Group 15

1 In the **Model Builder** window, click **1D Plot Group 15**.

2 In the **Settings** window for **1D Plot Group**, locate the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `x/h_th = 4.611`.

5 Locate the **Plot Settings** section.

6 Select the **y-axis label** check box. In the associated text field, type `u (m/s)`.

7 Locate the **Axis** section. Select the **Manual axis limits** check box.

8 In the **x minimum** text field, type `-0.1`.

9 In the **x maximum** text field, type `1.1`.

10 In the **y minimum** text field, type `-80`.

11 In the **y maximum** text field, type `320`.

12 Locate the **Grid** section. Select the **Manual spacing** check box.

13 In the **x spacing** text field, type `0.05`.

14 In the **y spacing** text field, type `20`.

15 In the **1D Plot Group 15** toolbar, click **Plot**.

1D Plot Group 16

1 Right-click **Results>1D Plot Group 15** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Line 2D 2**.

4 Locate the **Title** section. In the **Title** text area, type `x/h_th = 6.340`.

Line Graph 1

1 In the **Model Builder** window, expand the **1D Plot Group 16** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

3 In the **Expression** text field, type `y/0.066`.

Line Graph 2

1 In the **Model Builder** window, click **Line Graph 2**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `u_at6340(y/0.066)`.

4 Locate the **x-Axis Data** section. In the **Expression** text field, type `y/0.066`.

5 In the **1D Plot Group 16** toolbar, click **Plot**.