Created in COMSOL Multiphysics 6.1

Supersonic Air-to-Air Ejector

Introduction

This application models compressible turbulent gas flow in a supersonic air ejector using the High Mach Number Flow interface in COMSOL Multiphysics. Ejectors are simple mechanical components used to induce a secondary flow by momentum and energy transfer from a high-velocity primary jet. The high-energy fluid (primary flow) passes through a convergent-divergent nozzle and reaches supersonic conditions. After exiting the nozzle, it interacts with the secondary flow which is accelerated through an entrainment-induced effect. The mixing between both flows takes place along a constant-area duct called the mixing chamber where complex interactions between the mixing layer and shocks can be observed. A diffuser is usually placed before the outlet to recover pressure and bring the flow back to stagnation.

Ejectors are used for a wide range of applications, including industrial refrigeration, vacuum generation, gas recirculation, and thrust augmentation in aircraft propulsion systems. Great efforts have been made to determine their optimum design and operating conditions, as well as how to describe the flow within them (Ref. 1).

This application models an ejector working with air in both the primary and secondary streams. The geometry and boundary conditions are based on Ref. 2, and Ref. 3. The items of interest are the primary and secondary mass flows, the static pressure distribution along the centerline of the ejector, and the resolution of the flow in the mixing region.

Model Definition

Figure 1 shows the geometry of the ejector. Its dimensions can be found in Table 1. A two-dimensional axisymmetric geometry is used to approximate the 3D geometry of the device and to reduce the size of the problem.

The flow velocity in the ejector is large enough to introduce significant variations in the density and temperature of the fluid, and the flow is governed by the fully compressible Navier-Stokes equations. Moreover, the Mach number is expected to be larger than one in the divergent section of the primary nozzle, as well as in the mixing chamber. Interaction between the boundary layers and mixing layers cause the deceleration from supersonic to subsonic flow to take place through a complex succession of shocks called shock train or pseudo-shock wave phenomenon (Ref. 4). Thus, the mesh has to be fine enough to accurately capture this phenomenon.

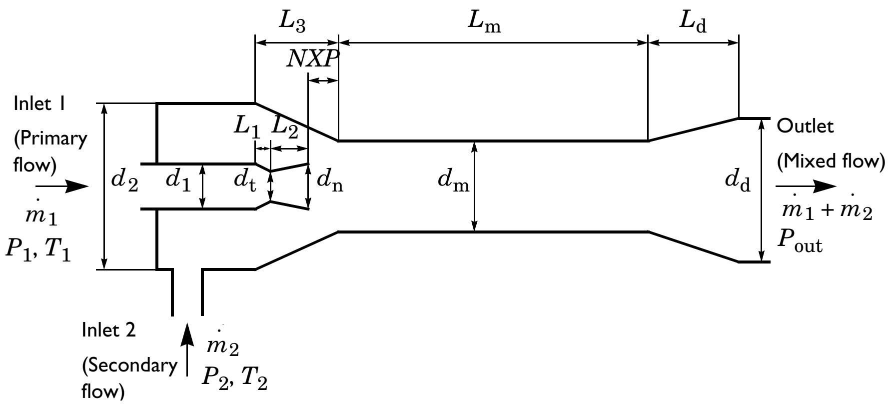


Figure 1: The geometry of the ejector.

TABLE 1: DIMENSIONS OF THE EJECTOR IN MM.

d_1	d_2	d_t	d_{nd}	d_m	d_d
16	160	8	12	24	51
L_1	L_2	L_3	L_m	L_d	NXP
7	23	90	240	70	15

The problem is modeled using the Favre-averaged Navier-Stokes equations and the standard k-ε turbulence model.

Both primary and secondary flows are air with a specific gas constant of 287 J/(kg·K) and a ratio of specific heats of 1.4. The dynamic viscosity and thermal conductivity of the air are computed from Sutherland's law.

BOUNDARY CONDITIONS

Inlet

The flow at the inlets is specified in terms of its total properties: $T_0 = 300$ K and $P_0 = 5$ atm for the primary flow, and $T_0 = 300$ K and $P_0 = 0.55$ atm for the secondary flow.

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.

The velocities at both inlets are unknown. However, they are expected to be very small compared to the velocities inside the nozzle and mixing chamber. The values that must be prescribed at the inlet are the total values of temperature and pressure, which define the energy of the flow. The Mach number can be set to 0 and will be determined by the characteristics based boundary condition at the inlet. This provides a good initial solution, but the total values of pressure and temperature may differ slightly. The solution can be improved if the problem is solved again setting the Mach number to the values computed by the characteristics based boundary condition at the inlets, which are approximately 0.14 and 0.01 for the primary and secondary inlets, respectively.

The inlet values for the turbulent kinetic energy, k, and the turbulent dissipation rate, ε, are approximated from the turbulent intensity, I_T, and turbulence length scale, L_T. Turbulent intensity is set by default to 0.05 (5%). The length scale can be approximated as 7% of the pipe diameter or hydraulic diameter. This is done automatically when Turbulence length scale is set to Geometry based.

For more background on this boundary condition, see the *CFD Module User's Guide*.

Outlet

The flow reaches supersonic conditions inside the ejector. However, it is expanded and decelerated along the mixing chamber and the diffuser, reaching subsonic conditions before being discharged into the atmosphere. The outlet is then subsonic with atmospheric static pressure. This is modeled using an Outlet node with the **Flow condition** set to subsonic.

Results and Discussion

The mass flows obtained are depicted in Table 2. The distributions of Mach number and velocity inside the ejector are depicted in Figure 2 and Figure 3. The primary flow is accelerated in the convergent section of the nozzle, reaching sonic conditions at the throat, and is expanded further in the divergent section. At the outlet of the primary nozzle, the secondary flow acts as an artificial wall for the primary flow. This leads to the formation of virtual nozzle throats, and a succession of expansion and compression waves can be observed in the region upstream of the mixing zone. Then, the flow decelerates along the constant-area duct and is brought back to stagnation in the diffuser. The region where both flows mix can be visualized by plotting the turbulent kinetic energy, see Figure 4.

Figure 5 plots the distribution of pressure along the centerline and walls of the mixing chamber. At the centerline of the duct, the flow successively changes from supersonic to subsonic flow via multiple shocks. However, this cannot be detected by wall pressure measurements because the surface pressures tend to be smeared out due to the dissipation in the boundary layer (see Ref. 4). The distribution of temperature is shown in Figure 6. Very low temperatures can be observed inside the device. This must be taken into account when designing an ejector, specially when working with two-phase flows. The results obtained correlate well with Ref. 2 and Ref. 3.

TABLE 2: MASS FLOWS.

$\dot{m}_1$	$\dot{m}_2$	$\dot{m}_{mixed}$
0.057 kg/s	0.036 kg/s	0.093 kg/s

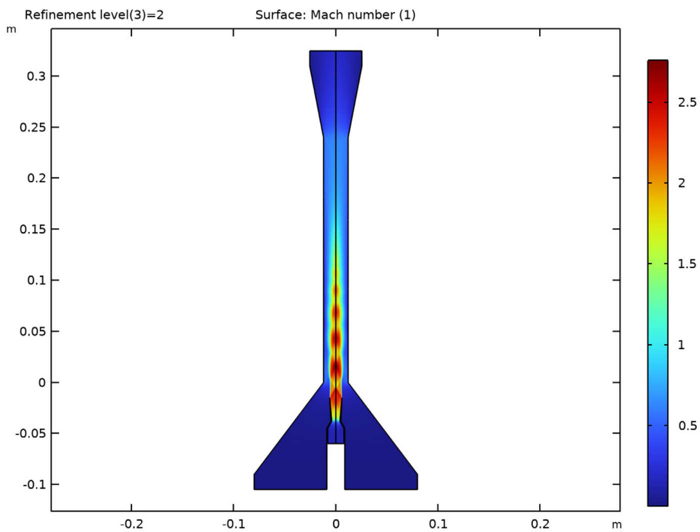


Figure 2: Mach number distribution.

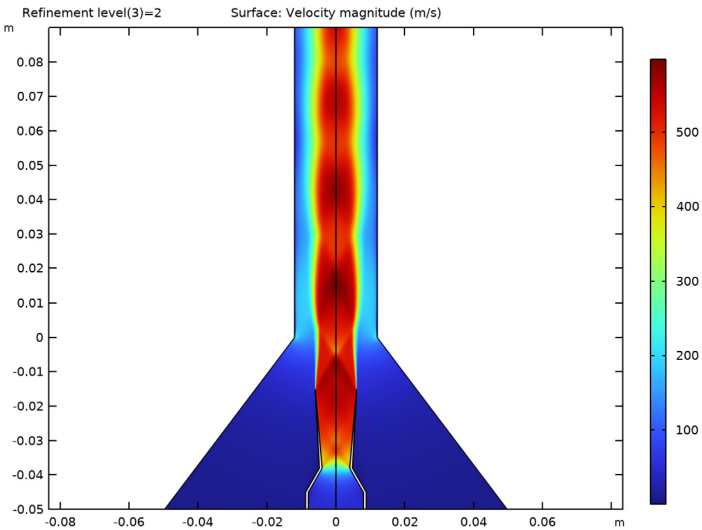


Figure 3: The distribution of velocity in the nozzle and the mixing chamber.

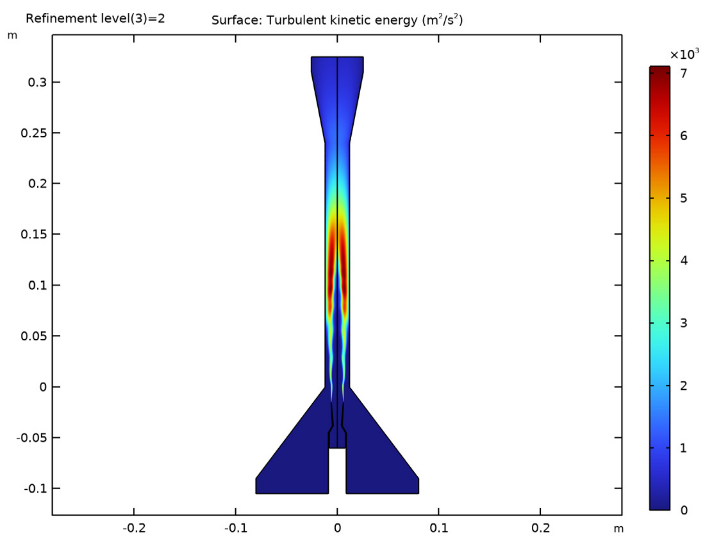


Figure 4: The distribution of turbulent kinetic energy. The mixing region is clearly identified.

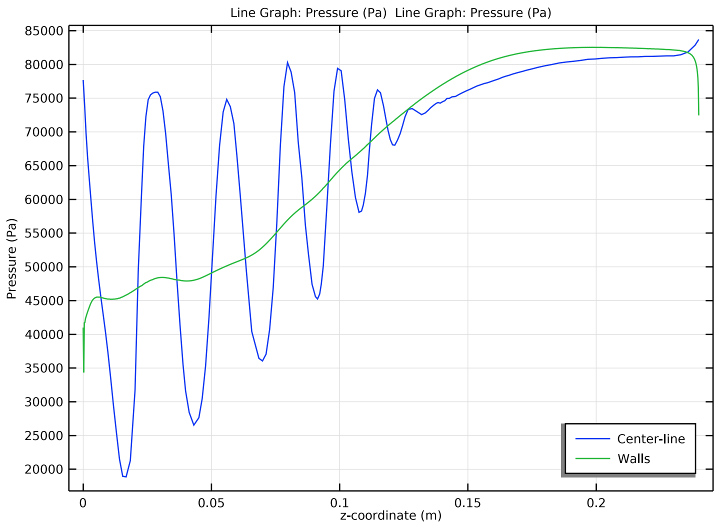


Figure 5: The distribution of pressure along the mixing chamber.

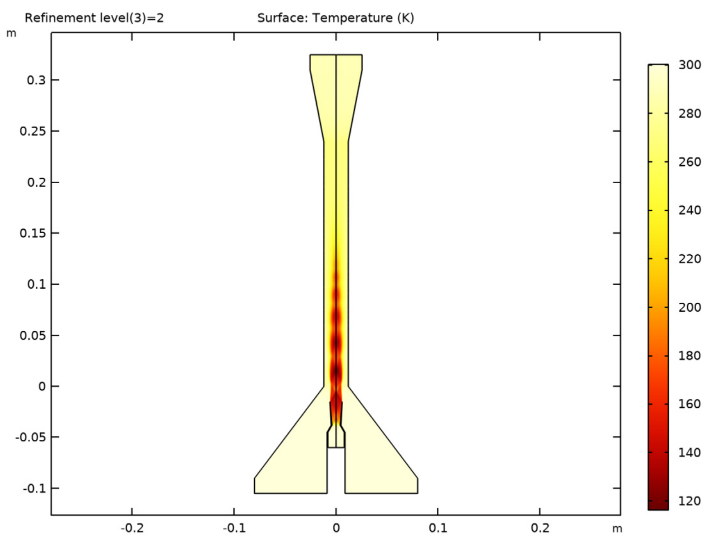


Figure 6: The distribution of temperature. Very low temperatures can be observed at the outlet of the nozzle. The temperature at the outlet of the ejector is lower than at both inlets.

Notes About the COMSOL Implementation

The present application is highly nonlinear and sensitive to the solution procedure. The mesh needed to capture the interaction between the shocks and the mixing layer, and to resolve the solution near the walls, is extremely fine. However, convergence may be hard to achieve with such a fine mesh unless a good enough initial solution is used. A way to overcome this is to first solve the problem on a coarse mesh and then refine it. The solution on the coarse mesh provides good initial values, but lacks accuracy in three important areas: wall resolution, capture of shocks, and resolution of the mixing layer. The adaptive mesh refinement feature can be used to overcome this. However, in order to fully resolve the mesh, a high element growth rate would be needed, potentially leading to convergence problems. An alternative option is to first refine manually both on the boundary layers and in the nozzle, and then to use the adaptive mesh refinement feature to resolve the mixing layer.

References

1. S. He, Y. Li, and R.Z. Wang, “Progress of Mathematical Modeling on Ejectors,” *Renew. Sustain. Energy Rev.*, vol. 13, pp 1760–1780, 2009.

2. Y. Bartosiewicz, Zine Aidoun, P. Desevaux, and Yves Mercadier, “Numerical and Experimental Investigations on Supersonic Ejectors,” *Int. J. of Heat and Fluid Flow*, vol. 26, pp 56–70, 2005.

3. P. Desevaux,A. Bouhangel, and E. Gavignet, “Flow Visualization in Supersonic Ejectors Using Laser Tomography Techniques,” *Int. J. of Refrigeration*, vol. 34, pp 1633–1640, 2010.

4. F. Gnani, H. Zare-Behtash, and K. Kontis, “Pseudo-shock Waves and their Interactions in High-speed Intakes,” *Progress in Aerospace Sciences,* vol. 82, pp 36–56, 2016.

Application Library path: `CFD_Module/High_Mach_Number_Flow/supersonic_ejector`

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 Axisymmetric**.

2 In the **Select Physics** tree, select **Fluid Flow>High Mach Number Flow>Turbulent Flow>High Mach Number Flow, k-ε (hmnf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

The model geometry is available as a parameterized geometry sequence in a separate MPH-file.

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `supersonic_ejector_geom_sequence.mph`.

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

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
P1	5[atm]	5.0663E5 Pa	Total pressure, primary flow
P2	0.55[atm]	55729 Pa	Total pressure, secondary flow
Pout	1[atm]	1.0133E5 Pa	Pressure, outlet
T1	300[K]	300 K	Total temperature, primary flow
T2	T1	300 K	Total temperature, secondary flow
Rs	287[J/kg/K]	287 J/(kg·K)	Specific gas constant
gamma	1.41	1.41	Ratio of specific heats
iso_diff	0	0	Isotropic diffusion

Add isotropic diffusion to improve convergence when computing the initial solution with a coarse mesh.

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>Stabilization**.

6 Click **OK**.

HIGH MACH NUMBER FLOW, K-ε (HMNF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **High Mach Number Flow, k-ε (hmnf)**.

2 In the **Settings** window for **High Mach Number Flow, k-ε**, click to expand the **Inconsistent Stabilization** section.

3 Find the **Heat equation** subsection. Select the **Isotropic diffusion** check box.

4 In the δ_{id} text field, type `iso_diff`.

5 Find the **Navier-Stokes equations** subsection. Select the **Isotropic diffusion** check box.

6 In the δ_{id} text field, type `iso_diff`.

Fluid 1

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

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

3 From the R_s list, choose **User defined**. In the associated text field, type `Rs`.

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

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

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

0	r
100	z

4 In the p text field, type `2[atm]`.

Inlet 1

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

2 In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Primary inlet**.

4 Locate the **Flow Properties** section. From the **Input state** list, choose **Total**.

5 In the $p_{0,\mathrm{tot}}$ text field, type `P1`.

6 In the $T_{0,\mathrm{tot}}$ text field, type `T1`.

7 In the Ma_0 text field, type `0.14`.

Inlet 2

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

2 In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Secondary inlet**.

4 Locate the **Flow Properties** section. From the **Input state** list, choose **Total**.

5 In the $p_{0,\mathrm{tot}}$ text field, type `P2`.

6 In the $T_{0,\mathrm{tot}}$ text field, type `T2`.

7 In the Ma_0 text field, type `0.01`.

Outlet 1

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

2 In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outlet**.

4 Locate the **Flow Condition** section. 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`.

The next step is to generate a coarse mesh to compute an initial solution.

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**.

4 Click **Build All**.

COMPONENT 1 (COMP1)

Generate a second mesh. Refine the boundary layer mesh and increase the mesh resolution on the walls and in the nozzle.

MESH 2

In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

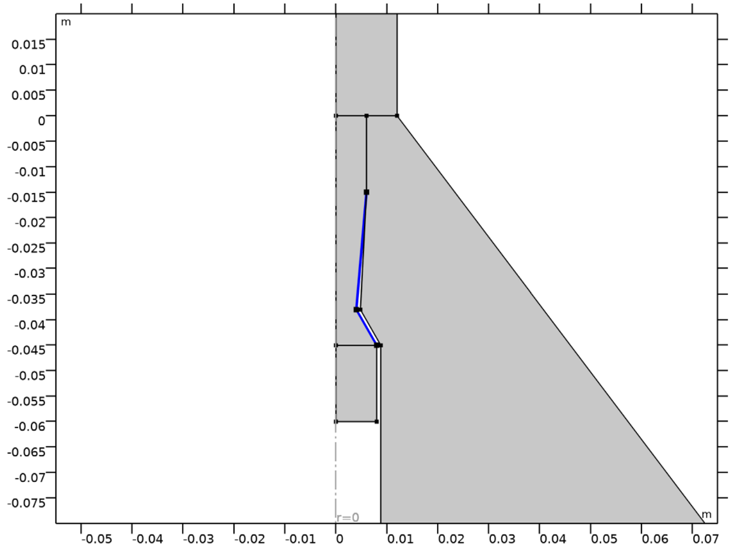
Size 1

1 Right-click **Mesh 2** 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 Boundaries 4 and 5 only (select the walls of the nozzle).



5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

6 From the **Predefined** list, choose **Extremely fine**.

Size 2

1 Right-click **Mesh 2** 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 From the **Selection** list, choose **Walls**.

5 In the list, choose **4** and **5**.

6 Click **Remove from Selection**.

7 Select Boundaries 6–9 and 11–15 only.

8 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

9 From the **Predefined** list, choose **Fine**.

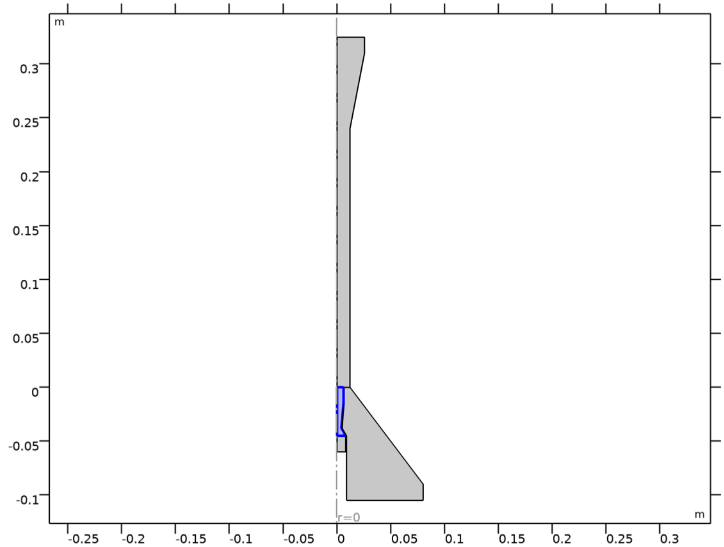
Size 3

1 Right-click **Mesh 2** and choose **Size**.

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

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

4 Select Domain 2 only.



5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

6 From the **Predefined** list, choose **Extremely fine**.

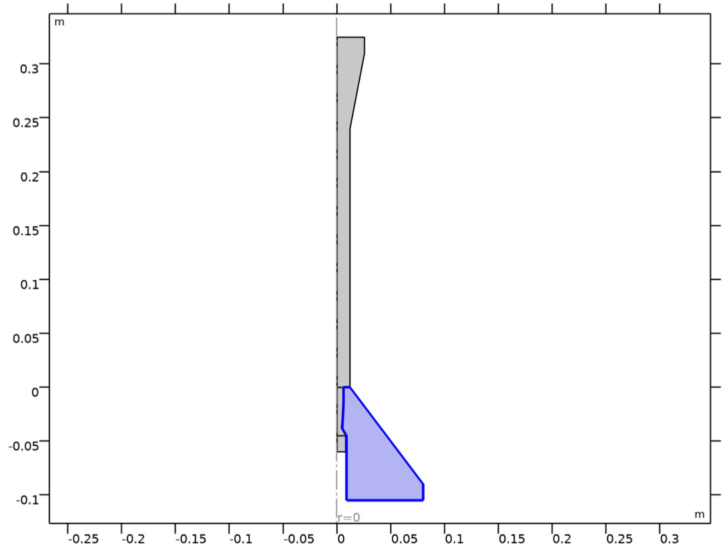
Size 4

1 Right-click **Mesh 2** and choose **Size**.

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

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

4 Select Domain 3 only.



5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

6 From the **Predefined** list, choose **Coarser**.

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Calibrate for** list, choose **Fluid dynamics**.

4 From the **Predefined** list, choose **Fine**.

Corner Refinement 1

1 In the **Mesh** toolbar, click **More Attributes** and choose **Corner Refinement**.

2 In the **Settings** window for **Corner Refinement**, locate the **Boundary Selection** section.

3 Click to select the **Activate Selection** toggle button.

4 From the **Selection** list, choose **Walls**.

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Boundary Layers 1

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

Boundary Layer Properties

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

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

3 From the **Selection** list, choose **Walls**.

4 In the list, choose **4**, **5**, and **8**.

5 Click **Remove from Selection**.

6 Select Boundaries 6, 7, 9, and 11–15 only.

7 Locate the **Layers** section. In the **Number of layers** text field, type `5`.

8 From the **Thickness specification** list, choose **First layer**.

9 In the **Thickness** text field, type `5e-5`.

Boundary Layer Properties 1

1 In the **Mesh** toolbar, click **More Attributes** and choose **Boundary Layer Properties**.

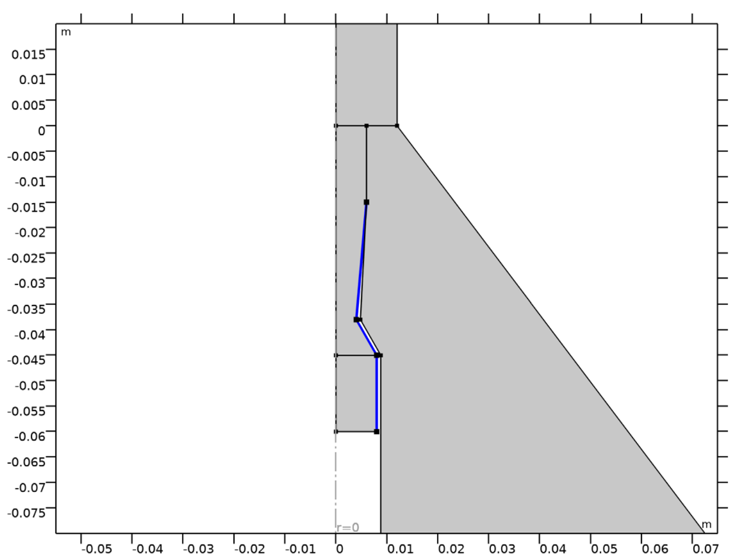
2 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

3 In the **Number of layers** text field, type `10`.

4 From the **Thickness specification** list, choose **First layer**.

5 In the **Thickness** text field, type `1e-5`.

6 Select Boundaries 4, 5, and 8 only (select the walls of both the primary inlet and the nozzle).



7 Click **Build All**.

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 **Mesh Selection** section.

3 In the table, enter the following settings:

Component	Mesh
Component 1	Mesh 1

Edit the study step to solve the model in two steps in order to increase the robustness of the solution procedure. First, solve for P_2=P_{out} (no adverse gradient of pressure), and then solve the problem again adding the adverse gradient of pressure (P_2 smaller than P_{out}). Disable the continuation solver since it is mainly suitable for linear applications.

4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
P2 (Total pressure, secondary flow)	`Pout 0.55[atm]`	`Pa`

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
iso_diff (Isotropic diffusion)	`0.5 0`	

9 From the **Run continuation for** list, choose **No parameter**.

10 From the **Reuse solution from previous step** list, choose **Yes**.

Solve the problem with a finer mesh, and use the adaptive mesh refinement feature to resolve the mixing layer. The solution of study step 1 is used as the initial guess.

Stationary 2

1 In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Mesh Selection** section.

3 In the table, enter the following settings:

Component	Mesh
Component 1	Mesh 2

4 Click to expand the **Adaptation and Error Estimates** section. From the **Adaptation and error estimates** list, choose **Adaptation and error estimates**.

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

Solution 1 (sol1)

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

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

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 2>Segregated 1** node, then click **Flow variables u, p, T**.

4 In the **Settings** window for **Segregated Step**, locate the **General** section.

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

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Stationary Solver 2** click **Adaptive Mesh Refinement**.

7 In the **Settings** window for **Adaptive Mesh Refinement**, locate the **General** section.

8 Clear the **Allow coarsening** check box.

9 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Stationary Solver 2>Segregated 1** click **Turbulence variables**.

10 In the **Settings** window for **Segregated Step**, locate the **General** section.

11 From the **Linear solver** list, choose **AMG, turbulence variables (hmnf)**.

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

Now, use Evaluation Group to compute individual mass flows and the difference between inlet and outlet mass flows to verify the mass conservation.

RESULTS

Evaluation mass flow group

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Evaluation mass flow group` in the **Label** text field.

Line Integration 1

1. Right-click **Evaluation mass flow group** and choose **Integration>Line Integration**.
2. In the **Settings** window for **Line Integration**, locate the **Expressions** section.
3. In the table, enter the following settings:

Expression	Unit	Description
hmnf.rho*w	kg/s	

Line Integration 2

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

Line Integration 3

In the **Model Builder** window, right-click **Line Integration 2** and choose **Duplicate**.

Primary mass flow

1. In the **Settings** window for **Line Integration**, type `Primary mass flow` in the **Label** text field.
2. Select Boundary 2 only.
3. Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
hmnf.rho*w	kg/s	Primary mass flow

Secondary mass flow

1. In the **Model Builder** window, under **Results>Evaluation mass flow group** click **Line Integration 2**.
2. In the **Settings** window for **Line Integration**, type `Secondary mass flow` in the **Label** text field.
3. Select Boundary 10 only.
4. Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
hmnf.rho*w	kg/s	Secondary mass flow

Mixed mass flow

1. In the **Model Builder** window, under **Results>Evaluation mass flow group** click **Line Integration 3**.
2. In the **Settings** window for **Line Integration**, type `Mixed mass flow` in the **Label** text field.

3 Select Boundary 3 only.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`hmnf.rho*w`	`kg/s`	`Mixed mass flow`

Now, write the expression and evaluate the difference between inlet (**int1+int2**) and outlet (**int3**).

Evaluation mass flow group

1 In the **Model Builder** window, click **Evaluation mass flow group**.

2 In the **Settings** window for **Evaluation Group**, locate the **Transformation** section.

3 From the **Transformation type** list, choose **General**.

4 In the **Expression** text field, type `int1+int2-int3`.

5 Select the **Keep child nodes** check box.

6 In the **Column header** text field, type `Difference between Inflow and Outflow`.

7 In the **Evaluation mass flow group** toolbar, click **Evaluate**.

Check if the average of the Mach number at the primary inlet is similar to the value defined in **Inlet 1**.

Mach number, primary inlet

1 In the **Results** toolbar, click **More Derived Values** and choose **Average>Line Average**.

2 In the **Settings** window for **Line Average**, type `Mach number, primary inlet` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Adaptive Mesh Refinement Solutions 1 (sol3)**.

4 From the **Parameter selection (Refinement level)** list, choose **Last**.

5 Locate the **Selection** section. From the **Selection** list, choose **Primary inlet**.

6 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`hmnf.Ma`	`1`	`Mach number`

7 Click **Evaluate**.

Check if the average of the Mach number at the secondary inlet is similar to the value defined in **Inlet 2**.

Mach number, secondary inlet

1 In the **Results** toolbar, click **More Derived Values** and choose **Average>Line Average**.

2 In the **Settings** window for **Line Average**, type `Mach number, secondary inlet` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Adaptive Mesh Refinement Solutions 1 (sol3)**.

4 From the **Parameter selection (Refinement level)** list, choose **Last**.

5 Locate the **Selection** section. From the **Selection** list, choose **Secondary inlet**.

6 Locate the **Expressions** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`hmnf.Ma`	1	`Mach number`

7 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

In case that the Mach number at one or both inlets diverge from the imposed values, the problem should be solved again using the new Mach numbers at the inlet.

RESULTS

Mirror 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.

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

3 From the **Dataset** list, choose **Study 1/Adaptive Mesh Refinement Solutions 1 (sol3)**.

4 Click **Plot**.

Plot the turbulent kinetic energy.

Turbulent Kinetic Energy

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Turbulent Kinetic Energy` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 1**.

Surface 1

1 Right-click **Turbulent Kinetic Energy** and choose **Surface**.

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

3 In the **Expression** text field, type `k`.

4 In the **Turbulent Kinetic Energy** toolbar, click **Plot** (see Figure 4).

Plot the temperature.

Temperature, 2D

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

2 In the **Settings** window for **2D Plot Group**, type `Temperature, 2D` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 1**.

Surface 1

1 Right-click **Temperature, 2D** and choose **Surface**.

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

3 In the **Expression** text field, type `T`.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Thermal>Thermal** in the tree.

6 Click **OK**.

7 In the **Temperature, 2D** toolbar, click **Plot** (see Figure 6).

Plot the evolution of pressure along the mixing chamber.

Center-line

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

2 In the **Settings** window for **Cut Line 2D**, type `Center-line` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Adaptive Mesh Refinement Solutions 1 (sol3)**.

4 Locate the **Line Data** section. In row **Point 2**, set **r** to `0`.

5 In row **Point 2**, set **z** to `L_mixing`.

6 Click **Plot**.

Wall

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

2 In the **Settings** window for **Cut Line 2D**, type `Wall` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Adaptive Mesh Refinement Solutions 1 (sol3)**.

4 Locate the **Line Data** section. In row **Point 1**, set **r** to `d_mixing/2`.

5 In row **Point 2**, set **r** to `d_mixing/2`.

6 In row **Point 2**, set **z** to `L_mixing`.

7 Click **Plot**.

1D Plot Group 10

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

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

3 From the **Position** list, choose **Lower right**.

Line Graph 1

1 Right-click **1D Plot Group 10** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Center-line**.

4 From the **Parameter selection (Refinement level)** list, choose **Last**.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `p`.

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

7 In the **Expression** text field, type `z`.

8 Click to expand the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`Center-line`

Line Graph 2

1 In the **Model Builder** window, right-click **1D Plot Group 10** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Wall**.

4 From the **Parameter selection (Refinement level)** list, choose **Last**.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `p`.

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

7 In the **Expression** text field, type `z`.

8 Locate the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
Walls

1D Plot Group 10

1 In the **Model Builder** window, click **1D Plot Group 10**.

2 In the **1D Plot Group 10** toolbar, click **Plot** (see Figure 5).

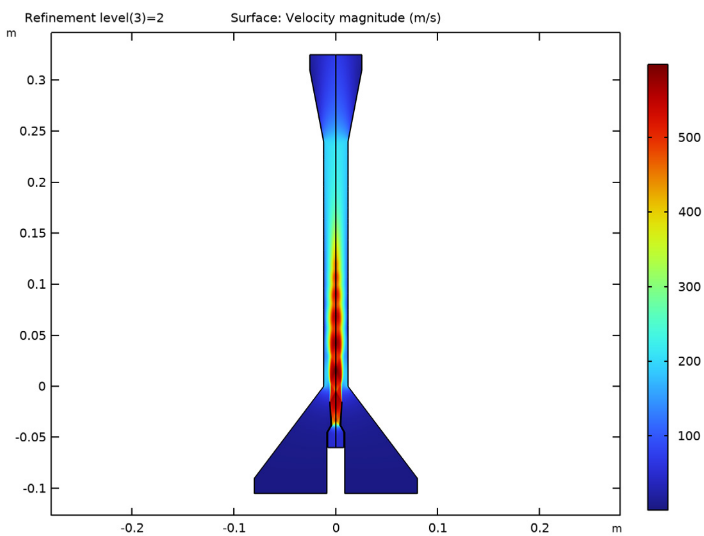
Velocity (hmnf)

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

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

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

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



Mach Number (hmnf)

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

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

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

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

5 Click the **Zoom Extents** button in the **Graphics** toolbar (see Figure 2).