Model created in COMSOL Multiphysics 6.4

Shock Diamonds from a Rectangular Nozzle

Introduction

In this example, the High Mach Number Flow, Low Reynolds Number k-ε interface is used to compute the compressible turbulent flow from a rectangular nozzle. Mesh refinement based on an estimation of the shock strength and the shear-layer strength is employed to achieve the appropriate resolution. The emerging turbulent supersonic jet has a shock diamond structure that attenuates along its axis due to growing turbulence viscosity. The asymmetry of the jet pattern between the minor and the major planes, and the entrainment of the surrounding low-speed flow are visualized.

Model Definition

High-speed turbulent flows occur in many cases of academic and practical interest: flow around supersonic and hypersonic vehicles as well as flow in rocket nozzles, ramjets, scramjets, and supersonic ejectors. This 3D model simulates a turbulent air flow from a supersonic nozzle into a large region with slow coflow.

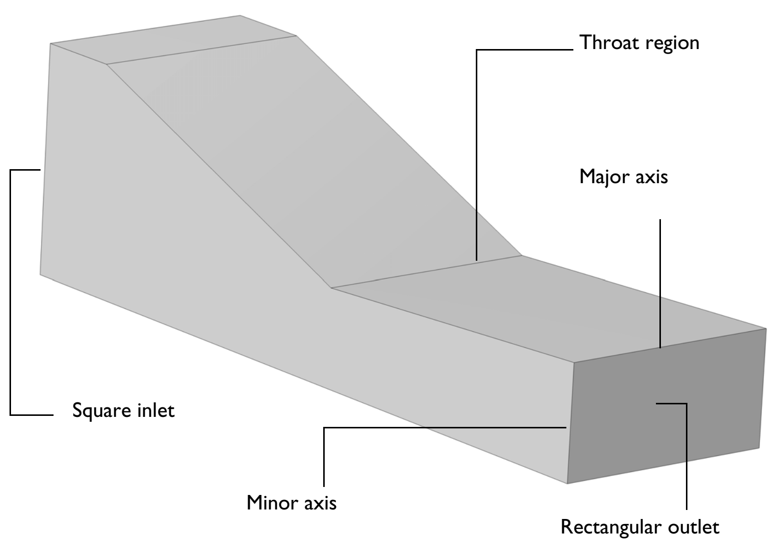

Figure 1: The geometry of the rectangular nozzle (air domain).

The idea of the geometry of the nozzle was presented in Ref. 1. The exact dimensions are not followed here; for example, this model uses blunter edges of the nozzle exit. In this study, only a stationary solution is computed, so it is sufficient to consider one quarter of the whole geometry; Figure 1 shows the corresponding air domain inside the nozzle. The major axis (in the horizontal plane) has a constant width, while the minor axis (in the vertical plane) has a convergent–divergent form. The inlet-to-throat area ratio of the nozzle is $26/11$, while the exit-to-throat area ratio is $13/11$. According to quasi-one-dimensional isentropic relations for a perfect gas with $\gamma = 1.4$, the Mach number at the throat is unity, $\mathrm{Ma} = 1$, while $\mathrm{Ma} = 0.2545$ at the nozzle inlet and $\mathrm{Ma} = 1.5083$ at the nozzle exit. Also, the relations indicate that the ratio of the pressures at the inlet and exit of the nozzle is 3.5518. Assuming that the ambient pressure is 1 atm, a static pressure at the inlet p_{in} lower than 3.55 atm (approximately, due to deviations from isentropicity and quasi-one-dimensionality) would result in overexpanded conditions, while p_{in} larger than that value would result in underexpanded conditions. Both situations are characterized by remarkable flow patterns consisting of shock and expansion waves enveloped by developing shear layers, which dominate further downstream, and slowly entraining flow.

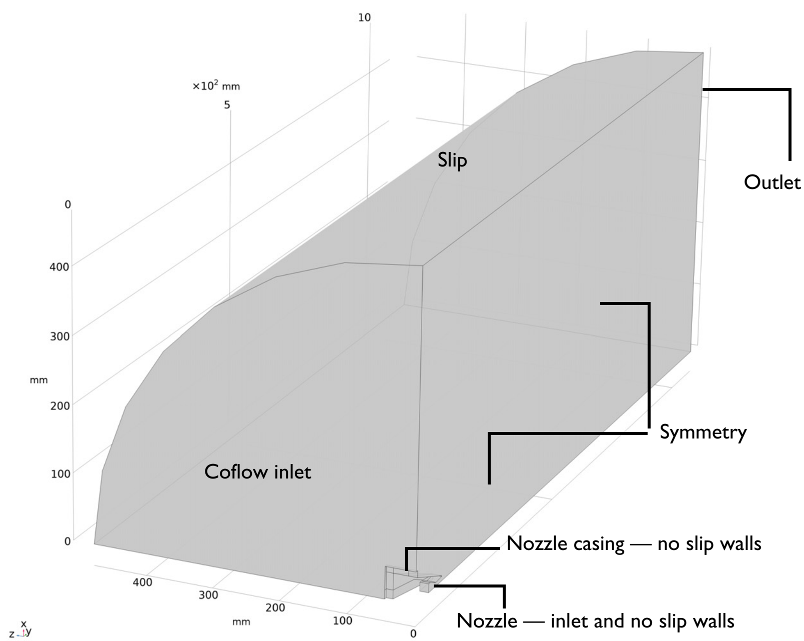

Figure 2: The whole computational geometry with boundary conditions.

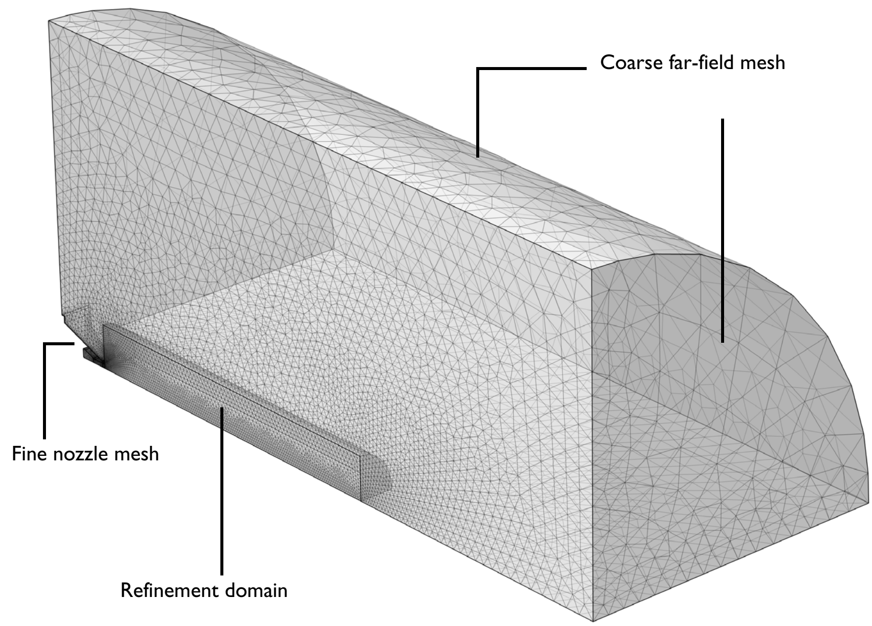

Figure 3: Mesh 2.

Figure 2 shows the boundary conditions for the model. At the nozzle inlet and at the low-speed co-inlet (coflow makes the computations more robust) total conditions are used. Here, a strongly underexpanded case with $p_{\mathrm{tot,in}} = 10$ atm is analyzed. The total pressure at the inlet is $T_{\mathrm{tot,in}} = 300$ K. The computational domain is large enough to ensure that the boundaries have little influence on the jet and to capture the entrainment correctly.

Implementation in COMSOL Multiphysics

Study 1, using the relatively coarse Mesh 1, ramps up the inlet total pressure from $p_{\mathrm{tot,in}} = 1.01$ atm to $p_{\mathrm{tot,in}} = 10$ atm in 10 steps. Each step takes 7 iterations only, since attaining convergence of those intermediate solutions is not of interest. Notice that the number of steps and the iterations per step can be varied depending on the convergence sensitivity on a particular computer architecture. Then, the mesh shown in Figure 3 and Figure 4 is built. Mesh 2 is sufficiently fine to resolve the evolution within the nozzle and in its close proximity. The large far-field region, which surrounds the nozzle and the jet regions, is kept coarse. The jet region is meshed quite well but not sufficiently well to capture shock waves, and is called the *Refinement domain*.

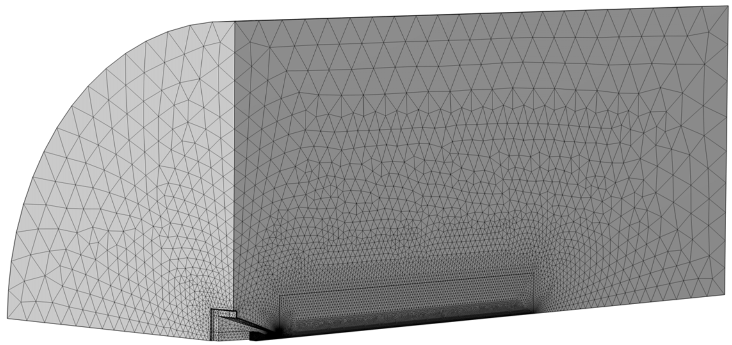

Figure 4: Mesh 2 from another view.

Indeed, without prior knowledge of the shock-wave pattern, an excessively fine mesh that covers most of the jet region would be required to guarantee resolution of the shocks. To avoid this, a shock indicator from Ref. 2 is employed by the adaptive mesh refinement strategy, which is limited to the refinement domain. Study 2 uses the last solution of Study 1 as the initial value, and starts by obtaining a solution on Mesh 2. This refinement level-0 solution is used to perform the error estimation based on a modified (to include the shear layers) indicator, and a level-1 adapted mesh is built. The refinement level-1 solution obtained on this mesh is used to perform the error estimation based on the shock indicator. A level-2 adapted mesh is produced and the refinement level-2 solution is obtained, which is considered as the final result that is presented below in 2D and 3D images (since further refinements using this form of the shock indicator lead to excessively large number of elements). Figure 5 shows the mesh evolution through the refinements.

Study 2 uses nondefault settings in the segregated solver that were found to be more suitable for this particular problem compared to the default settings.

The results in the far-field region with a coarse mesh are not very reliable. Nevertheless, it is expected that entrainment by that part of the jet which is inside the refinement domain would be captured correctly.

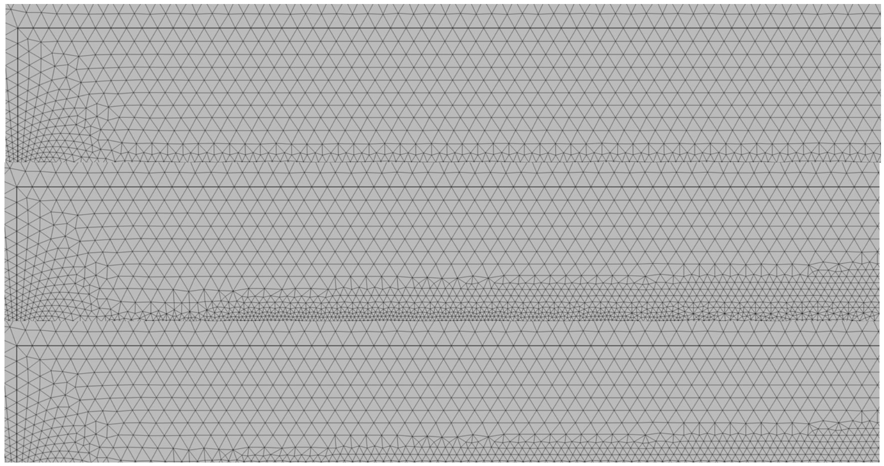

Figure 5: Adaptive mesh refinement on the Refinement domain: top — original Mesh 2, Middle — shear layers and shock regions refined (Level 1 Adapted Mesh); bottom — shock regions refined (Level 2 Adapted Mesh).

The above shock error indicator focuses mostly on the strongest shocks. An improved approach is needed to arrive at a more homogeneous refinement of all the shocks.

The details of the implementation for the High Mach Number Flow, Low Reynolds Number k-ε interface and of the Adaptation and Error Estimates can be found in the *CFD Module User's Guide*; see the sections Theory for the High Mach Number Flow Interfaces, Theory for the Turbulent Flow Interfaces, Adaptive Mesh Refinement (Stationary and Eigenvalue Adaptation), and Error Estimation — Theory and Variables.

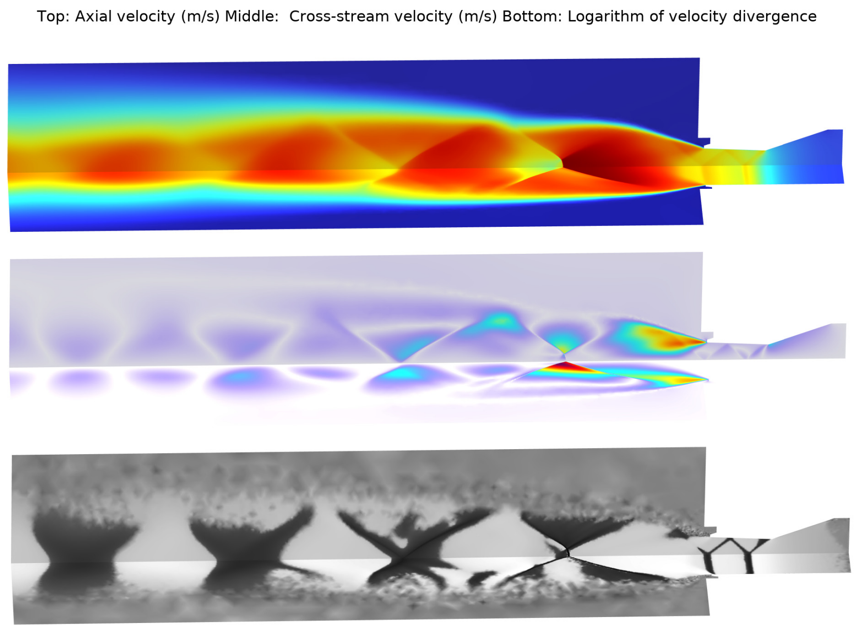

Figure 6: Asymmetry between the vertical plane (minor axis) and horizontal plane (major axis). Color legends are found in the model file. Bottom: light gray regions — expansion regions; dark regions — compression regions.

The three-dimensional perspective of the jet, Figure 6, demonstrates the asymmetry between the vertical (minor axis) and horizontal (major axis) planes of the model. The Mach disk, possessed by the first shock diamond, can be seen on the "convergent-divergent plane" (vertical, or minor axis) only. It is not a real normal shock, since the flow remains supersonic on passing it. Very strong maximum of the cross-stream velocity is observed in the "constant-width plane" (horizontal, or major axis) near the Mach disk. On the bottom part of Figure 6 the regions of expansion and compression are clearly visible. Shock waves are formed already in the nozzle during the expansion (thus violating isentropic conditions and quasi one-dimensionality). The first and the second shock diamonds reveal striking asymmetry, but the subsequent shock diamonds become more and more symmetric.

The asymmetry between the planes is also illustrated by the two-dimensional perspective in Figure 7. The structure of the shock diamonds becomes more symmetric along the axis of the jet, it also becomes smeared out. The vorticity generated by the structure adherent to the Mach disk is comparable to the maximum vorticity in the shear layers, but is very localized. Observe that at the nozzle exit the temperature is already as low as 200 K, and that a very low temperature, 83 K, is achieved just upstream of the Mach disk.

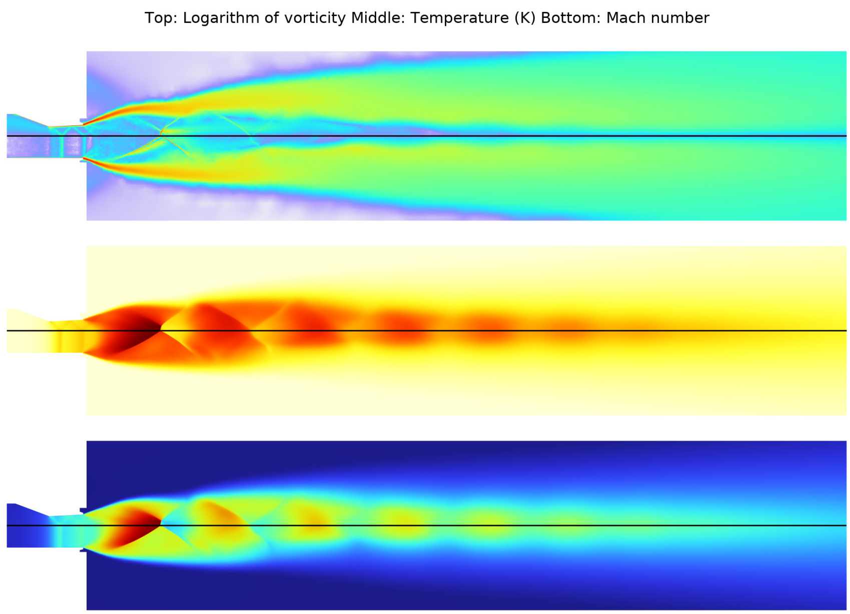

Figure 7: Asymmetry between the vertical and horizontal planes — planar perspective. Above the dividing lines Q vertical (minor axis) surface; below the dividing lines — horizontal (major axis) surface.

Inspect the model file to observe how the details of the solution on Figure 6 and Figure 7 change when going from Refinement level 0 to Refinement level 1 to Refinement level 2. For example, the first and the second shock diamonds are much sharper in Refinement 2 than in Refinement 1, while Refinement 0 produces rather diffuse shock diamonds.

Figure 8 demonstrates the evolution of Mach number along the jet axis. Mach number at the nozzle exit is close to the above estimated value. Strong expansion follows (since the conditions are underexpanded), so that $\mathrm{Ma} = 3.6$ is reached, then a Mach disk occurs with $\mathrm{Ma} = 1.1$ behind it. Several more expansion-compression cycles happen. Notice that adaptive refinements “pull” the pattern closer to the nozzle compared to the No refinement solution, and that difference between Refinement 1 and Refinement 2 is insignificant. This would indicate the mesh convergence, however keep in mind that further refinement would produce sharper peaks and troughs in Figure 8.

Figure 9 plots the evolution of the decimal logarithm of the velocity divergence along the jet axis. It is clearly seen that at least two shock diamonds (in the tail of the wave dominated portion of the jet) are “lost” due to refinements. Indeed, it can be shown that the refinements consistently make turbulent viscosity higher, leading to faster smearing of the shock diamond structure. Figure 10 plots the evolution of pressure along the jet axis and the attenuation observed is consistent with the previous figures. A very low pressure, 0.117 atm, is achieved just upstream of the Mach disk (at the maximum Mach number), while the peak pressure just behind the Mach disk is 2.9 atm, which is higher than at the nozzle exit, 2.3 atm.

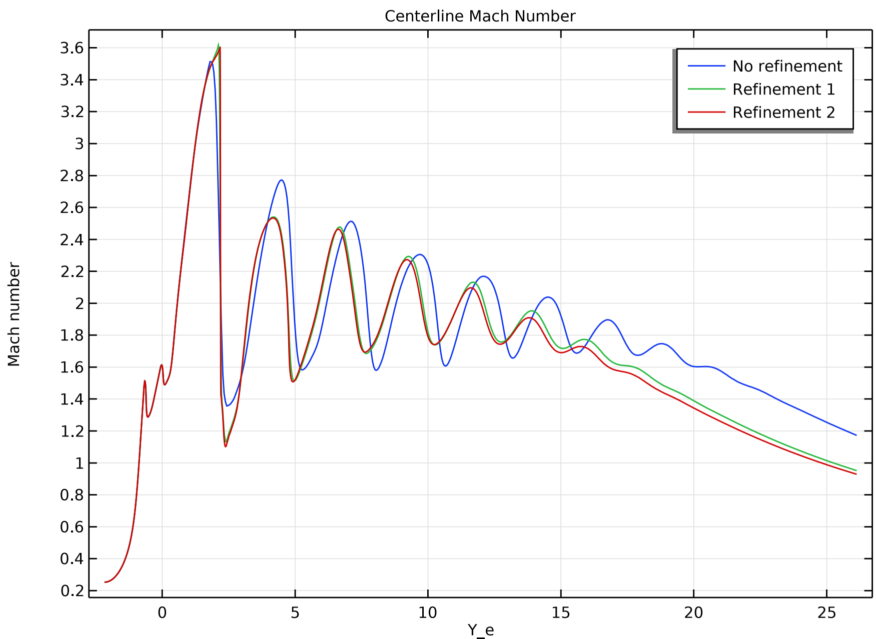

Figure 8: Mach number along the jet axis. Y_e is the distance along the jet axis normalized by the equivalent diameter of the nozzle D_e. Zero point is set exactly at the nozzle exit. No refinement corresponds to solution on Mesh 2.

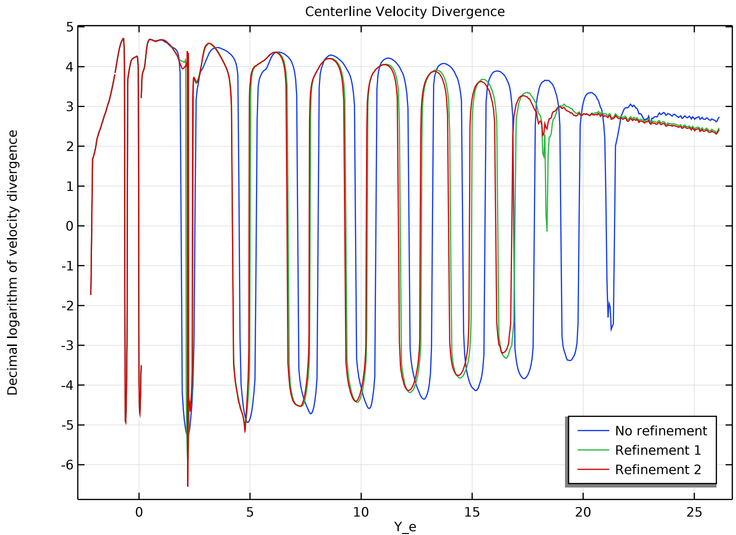

Figure 9: Velocity divergence at the axis of the jet. See description of Figure 8.

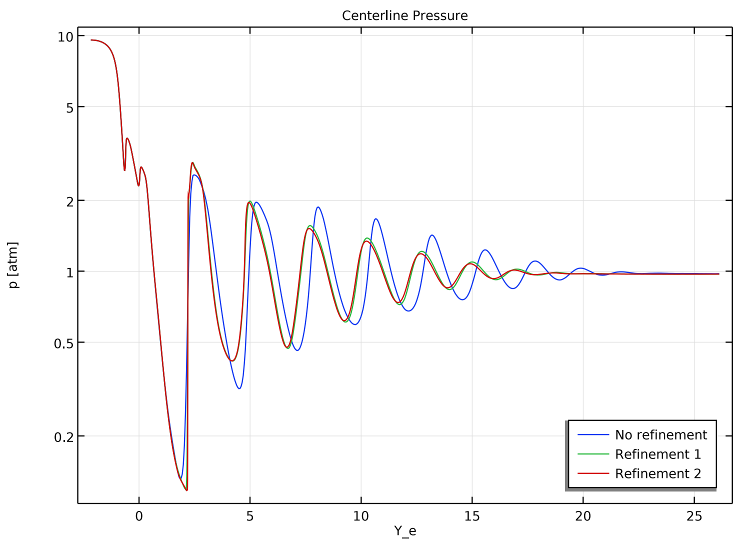

Figure 10: Pressure at the axis of the jet. See description of Figure 8.

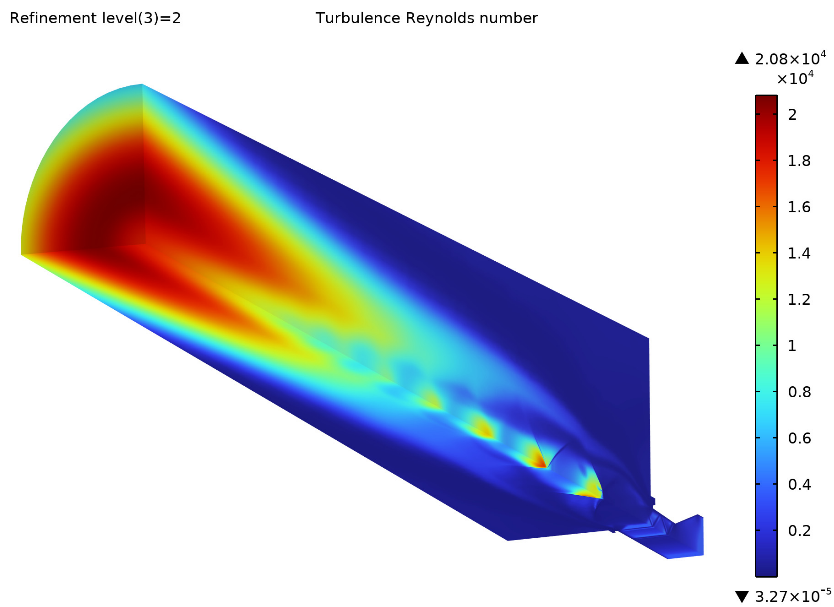

Figure 11: Turbulence Reynolds number at the surfaces of the Refinement domain.

Figure 11 shows the turbulence Reynolds number, equal to the ratio of the turbulence and the dynamics viscosities. The picture confirms that when the turbulence generated by the shear layers becomes significant, it destroys the shock-diamond pattern. Notice that the shock diamonds are able to generate quite high turbulence with large values of the turbulence Reynolds number, but those “turbulent patches” are rather localized and quickly attenuate. Figure 11 essentially illustrates the main reason of quite fast transition from a supersonic jet dominated by inviscid wave phenomena (compression and expansion waves) and small influence of turbulence to a “normal” subsonic jet dominated by the turbulence effects.

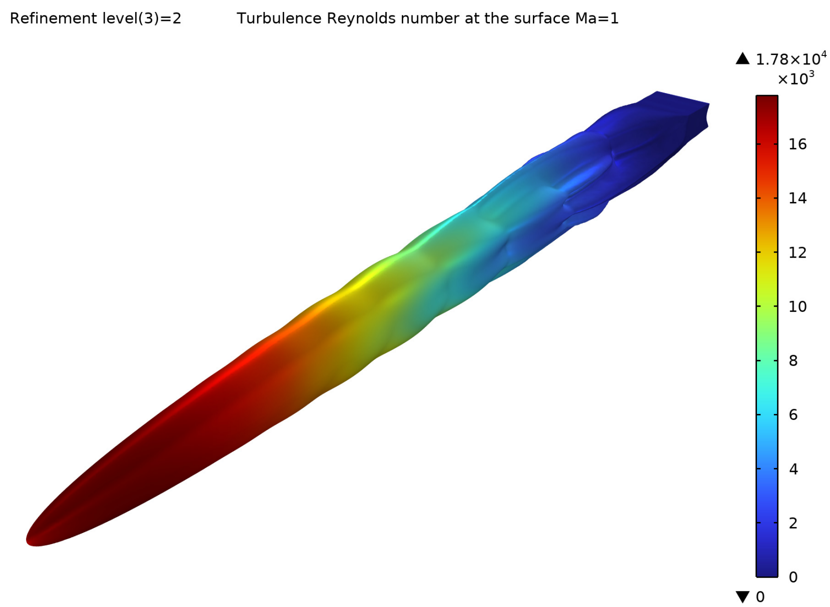

Figure 12: Supersonic jet envelope — the enveloping surface with $\mathrm{Ma} = 1$.

Figure 12 illustrates the supersonic jet envelope. It is remarkable that even local subsonic patches inside the envelope are absent, so that flow can be distinctly subdivided into subsonic and supersonic regions. This further confirms that there are no normal shock waves inside the jet. After the shock diamond pattern becomes smeared out by the growing viscosity, the flow of this supersonic part becomes weakly expanding, which is confirmed by Figure 9. The pronounced wave pattern in the beginning of the envelope is highly asymmetric and full of details, but when the wave pattern disappears, the cross sections of the envelope quickly approach circular profiles.

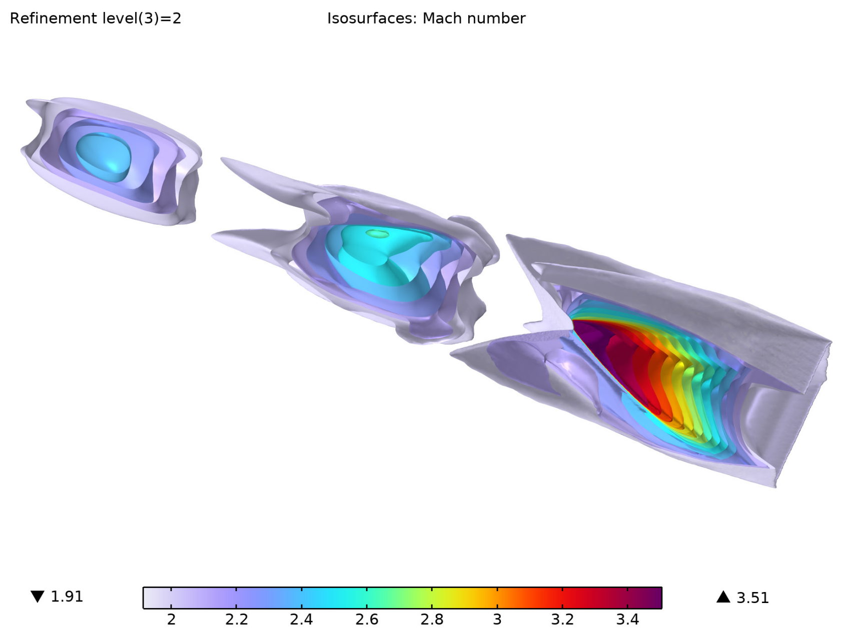

Figure 13: Shock diamond isosurfaces $\mathrm{Ma} = \mathrm{const}$*; the set at the image is* $[2{,}0.125{,}3.5]$.

Figure 13 shows the Mach number isosurfaces of the first three shock diamonds. The structure is very deformed, it possesses very elongated and sharp features. This is in sharp contrast with Refinement level-0 version of the image. Look into the model file to see how the choice of the Refinement level effects the isosurfaces (try also different sets of isosurfaces). Remark that the resolved shock diamond structure looks, in some sense, more irregular versus the underresolved case. Thus, a simple visual inspection may help to estimate the chosen mesh accuracy in a particular computation of a shock diamond wave pattern.

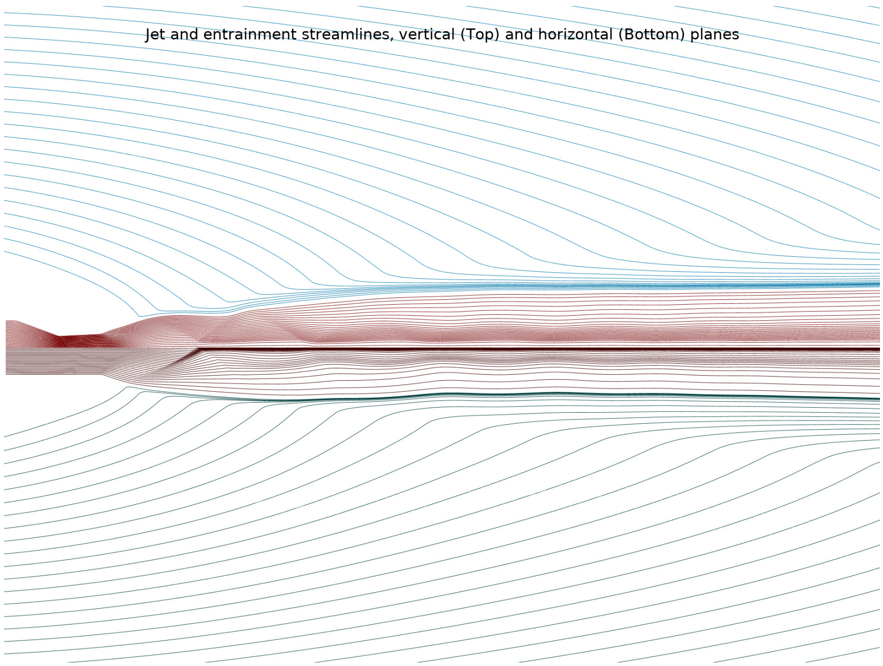

Figure 14: The streamline pattern. Top - minor axis (convergent-divergent cross section of the nozzle) vertical plane, bottom - major axis (constant width cross section of the nozzle) horizontal plane.Bright colors - jet flow, calm colors - entrainment flow.

Figure 14 demonstrates the streamlines in the jet and the entrained flow. Quite remarkably, the jet flow has a strong tendency to concentrate in the near-axis region. An especially sharp deviation is revealed in the horizontal plane, where nearly half of all the streamlines very quickly are collected in a very thin region near the axis. This strong deviation of the streamlines in the horizontal plane corresponds to very high cross-stream velocity in the middle of Figure 6 and is caused by the Mach disk. Zoom out Figure 14 in the model file and notice that the initial waviness of the jet/entrainment boundary evolves into straight lines. This again illustrates the transition from the pattern with dominant shock diamonds to the pattern of the turbulence dominated subsonic jet.

Summary and Outlook

To summarize, the COMSOL Multiphysics computations are able to reveal main features of the high-speed jet flow with highly asymmetric structure due to the nozzle geometry, both in the supersonic and subsonic portions. The importance of a proper refinement strategy to avoid excessively large meshes while still being able to resolve shock waves as well as shear layers is demonstrated.

Various nozzle geometries in underexpanded or overexpanded conditions can be analyzed and different approaches to refinement can be tested.

Only consistent stabilization is used in this study, and the target CFL number for pseudo time stepping is 10000. In the case of a very sensitive model, inconsistent stabilization or a lower target CFL number can be tried to achieve a stationary solution, although the relevance of such a solution must be justified in each particular case.

References

1. K. Bhide, K. Siddappaji, and S. Abdallah "Influence of Fluid–Thermal–Structural Interaction on Boundary Layer Flow in Rectangular Supersonic Nozzles," *Aerospace*, vol. 5, p. 33, 2018; doi.org/10.3390/aerospace5020033; copyright 2018 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (creativecommons.org/licenses/by/4.0/).

2. D. Moro, N.C. Nguyen, and J. Peraire, "Dilation-based shock capturing for high-order methods," *Int. J. Numer. Methods Fluids*, vol. 82, no. 7), pp. 398–416, 2016.

Application Library path: `CFD_Module/High_Mach_Number_Flow/rectangular_nozzle`

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** > **Turbulent Flow** > **High Mach Number Flow, Low Reynolds Number k-ε** (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**.

GEOMETRY 1

1 Click the **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog, click **Select All**.

3 Click **OK**.

GLOBAL DEFINITIONS

Parameters 1

1 From the **File** menu, choose **Save As**.

2 Browse to a suitable folder, enter the filename `rectangular_nozzle.mph`, and then click **Save**.

3 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

4 In the **Settings** window for **Parameters**, locate the **Parameters** section.

5 Click **Load from File**.

6 Browse to the model's Application Libraries folder and double-click the file `rectangular_nozzle_parameters1.txt`.

Load two parameter files with geometric and physical parameters.

Parameters 2

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

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `rectangular_nozzle_parameters2.txt`.

DEFINITIONS

Before setting up the geometry, create three View nodes for later use in image export.

View 2

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

View 3

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

View 4

In the **Model Builder** window, right-click **View 3** and choose **Duplicate**.

GEOMETRY 1

It suffices to create a quarter of the full geometry due to symmetry.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Work Plane Major Axis

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

2 In the **Settings** window for **Work Plane**, type `Work Plane Major Axis` in the **Label** text field.

3 Locate the **Plane Definition** section. From the **Plane** list, choose **zy-plane**.

Work Plane Major Axis (wp1) > Plane Geometry

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

2 In the **Settings** window for **Plane Geometry**, locate the **Visualization** section.

3 Select the **View work plane geometry in 3D** checkbox.

Work Plane Major Axis (wp1) > Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `w_ba_ma/2+w_a`.

4 In the **Height** text field, type `h_tot`.

Work Plane Major Axis (wp1) > Rectangle 2 (r2)

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

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

3 In the **Width** text field, type `w_ma/2`.

4 In the **Height** text field, type `h_tot`.

Work Plane Major Axis (wp1) > Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `rectangular_nozzle_polygon1.txt`.

Work Plane Major Axis (wp1) > Difference 1 (dif1)

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

2 Select the object **r1** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the objects **pol1** and **r2** only.

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

Work Plane Minor Axis

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

2 In the **Settings** window for **Work Plane**, type `Work Plane Minor Axis` in the **Label** text field.

Work Plane Minor Axis (wp2) > Plane Geometry

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

2 In the **Settings** window for **Plane Geometry**, locate the **Visualization** section.

3 Select the **View work plane geometry in 3D** checkbox.

Work Plane Minor Axis (wp2) > Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `w_ba_mi/2+w_a`.

4 In the **Height** text field, type `h_tot`.

Work Plane Minor Axis (wp2) > Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `rectangular_nozzle_polygon2.txt`.

Work Plane Minor Axis (wp2) > Polygon 2 (pol2)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `rectangular_nozzle_polygon3.txt`.

Work Plane Minor Axis (wp2) > Difference 1 (dif1)

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

2 Select the object **r1** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the objects **pol1** and **pol2** only.

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

Extrude Major Axis

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

2 In the **Settings** window for **Extrude**, type `Extrude Major Axis` in the **Label** text field.

3 Locate the **General** section. From the **Work plane** list, choose **Work Plane Major Axis (wp1)**.

4 Select the object **wp1** only.

5 Locate the **Distances** section. In the table, enter the following settings:

Distances (mm)
`w_ba_mi/2+w_a`

6 Select the **Reverse direction** checkbox.

Extrude Minor Axis

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

2 In the **Settings** window for **Extrude**, type `Extrude Minor Axis` in the **Label** text field.

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

Distances (mm)
`w_ba_ma/2+w_a`

4 Click **Build All Objects**.

Union 1 (uni1)

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

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Union**, locate the **Selections of Resulting Entities** section.

4 Find the **Cumulative selection** subsection. Click **New**.

5 In the **New Cumulative Selection** dialog, type `Nozzle_mold` in the **Name** text field.

6 Click **OK**.

Delete Entities 1 (del1)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **uni1**, select Domains 4 and 5 only.

5 Click **Build All Objects**.

Linking Domain

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

2 In the **Settings** window for **Block**, type `Linking Domain` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `0.8*w_mi_t`.

4 In the **Depth** text field, type `0.2*(h_tot-h_mi_mt)`.

5 In the **Height** text field, type `0.6*w_ma`.

6 Locate the **Position** section. In the **y** text field, type `h_tot-0.1*(h_tot-h_mi_mt)`.

7 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

8 In the **New Cumulative Selection** dialog, type `Linking` in the **Name** text field.

9 Click **OK**.

Create the domain on which adaptive mesh refinement will be applied.

Work Plane, Refinement Cross Section

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

2 In the **Settings** window for **Work Plane**, type `Work Plane, Refinement Cross Section` in the **Label** text field.

3 Locate the **Plane Definition** section. From the **Plane** list, choose **zx-plane**.

4 In the **y-coordinate** text field, type `h_tot+0.1*(h_tot-h_mi_mt)`.

Work Plane, Refinement Cross Section (wp3) > Plane Geometry

1 In the **Model Builder** window, click **Plane Geometry**.
2 In the **Settings** window for **Plane Geometry**, locate the **Visualization** section.
3 Select the **View work plane geometry in 3D** checkbox.

Work Plane, Refinement Cross Section (wp3) > Ellipse 1 (e1)

1 In the **Work Plane** toolbar, click **Ellipse**.
2 In the **Settings** window for **Ellipse**, locate the **Size and Shape** section.
3 In the **a-semiaxis** text field, type `2*w_ma`.
4 In the **b-semiaxis** text field, type `5*w_mi_t`.
5 In the **Sector angle** text field, type `90`.

Refinement Domain

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.
2 In the **Settings** window for **Extrude**, type `Refinement Domain` in the **Label** text field.
3 Locate the **Distances** section. In the table, enter the following settings:

Distances (mm)
6*h_tot

4 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.
5 In the **New Cumulative Selection** dialog, type `Refinement` in the **Name** text field.
6 Click **OK**.

Cutting Domain

1 In the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for **Block**, type `Cutting Domain` in the **Label** text field.
3 Locate the **Size and Shape** section. In the **Width** text field, type `w_mi_b/2`.
4 In the **Depth** text field, type `h_mi_ml-(h_mi_mt-h_mi_ml)/4`.
5 In the **Height** text field, type `w_ma/2`.

Create a large computational domain to reduce the influence of boundary conditions on the jet and to obtain the correct amount of entrainment.

Work Plane, Computational Cross Section

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

2 In the **Settings** window for **Work Plane**, type `Work Plane, Computational Cross Section` in the **Label** text field.

3 Locate the **Plane Definition** section. From the **Plane** list, choose **zx-plane**.

Work Plane, Computational Cross Section (wp4) > Plane Geometry

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

Work Plane, Computational Cross Section (wp4) > 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 `sqrt(uc_wa_h*uc_wa_w)`.

4 In the **Sector angle** text field, type `90`.

Computational Domain

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

2 In the **Settings** window for **Extrude**, type `Computational Domain` in the **Label** text field.

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

Distances (mm)
`H_tot`

Difference 1 (dif1)

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

2 Select the objects **blk1**, **ext3**, and **ext4** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the objects **blk2** and **del1** only.

6 Click **Build All Objects**.

Work Plane 5 (wp5)

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 **zx-plane**.

4 In the **y-coordinate** text field, type `h_mi_mt-(h_mi_mt-h_mi_ml)/8`.

Partition Domains 1 (pard1)

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

2 On the object **dif1**, select Domain 2 only.

3 In the **Settings** window for **Partition Domains**, click **Build All Objects**.

Disable the analysis of the geometry as the remaining small geometric details can be kept.

4 In the **Model Builder** window, click **Geometry 1**.

5 In the **Settings** window for **Geometry**, locate the **Cleanup** section.

6 Clear the **Automatic detection of small details** checkbox.

Create predefined selections for later use.

DEFINITIONS

Inlet

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 In the **Label** text field, type `Inlet`.

5 Select Boundary 8 only.

Co-Inlet

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Co-Inlet` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 2 only.

Outlet

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Outlet` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 30 only.

No-Slip Wall, Coarse Mesh

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 In the **Label** text field, type `No-Slip Wall, Coarse Mesh`.

5 Select Boundaries 3, 5, 6, 33, 41, 42, 44, 45, and 47–51 only.

No-Slip Wall, Fine Mesh

1 In the **Definitions** toolbar, click **Difference**.

2 In the **Settings** window for **Difference**, type `No-Slip Wall, Fine Mesh` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog, select **Nozzle_mold** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference**, locate the **Input Entities** section.

8 Under **Selections to subtract**, click **Add**.

9 In the **Add** dialog, select **No-Slip Wall, Coarse Mesh** in the **Selections to subtract** list.

10 Click **OK**.

Slip Wall

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Slip Wall` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 4 only.

Symmetry Boundary

1 In the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for **Complement**, type `Symmetry Boundary` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

5 In the **Add** dialog, in the **Selections to invert** list, choose **Inlet**, **Co-Inlet**, **Outlet**, **No-Slip Wall, Coarse Mesh**, **No-Slip Wall, Fine Mesh**, and **Slip Wall**.

6 Click **OK**.

Nozzle 1

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Nozzle 1` in the **Label** text field.

3 Select Domain 2 only.

Nozzle 2

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Nozzle 2` in the **Label** text field.

3 Select Domains 3 and 4 only.

Nozzle+Linking+Refinement

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Nozzle+Linking+Refinement` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog, in the **Selections to add** list, choose **Nozzle 1**, **Nozzle 2**, and **Refinement**.

5 Click **OK**.

Refinement Edge

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Refinement Edge` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Edge**.

4 Select Edge 34 only.

Main-Region Boundaries

1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, type `Main-Region Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog, select **Nozzle+Linking+Refinement** in the **Input selections** list.

5 Click **OK**.

Vertical Plane

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Vertical Plane` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 9, 13, 17, and 25 only.

Vertical Extended Plane

1 Right-click **Vertical Plane** and choose **Duplicate**.

2 In the **Settings** window for **Explicit**, type `Vertical Extended Plane` in the **Label** text field.

3 Select Boundaries 9, 13, 17, 25, and 29 only.

Horizontal Plane

1 Right-click **Vertical Extended Plane** and choose **Duplicate**.

2 In the **Settings** window for **Explicit**, type `Horizontal Plane` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Remove from Selection**.

4 Click **Clear Selection**.

5 Select Boundaries 7, 11, 15, and 23 only.

Horizontal Extended Plane

1 Right-click **Horizontal Plane** and choose **Duplicate**.

2 In the **Settings** window for **Explicit**, type `Horizontal Extended Plane` in the **Label** text field.

3 Select Boundaries 1, 7, 11, 15, and 23 only.

Vertical+Horizontal Planes

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Vertical+Horizontal Planes` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Vertical Plane** and **Horizontal Plane**.

6 Click **OK**.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in** > **Air**.

4 Click the **Add to Component** button in the window toolbar.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

HIGH MACH NUMBER FLOW, LOW REYNOLDS NUMBER K-ε (HMNF)

1 In the **Settings** window for **High Mach Number Flow, Low Reynolds Number k-ε**, locate the **Physical Model** section.

2 In the T_{ref} text field, type `T_in`.

3 Click to expand the **Advanced Settings** section.

Fluid 1

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

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

3 From the k list, choose **From material**.

4 Locate the **Dynamic Viscosity** section. From the μ list, choose **From material**.

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 In the T text field, type `T_in`.

Inlet 1

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

Apply total conditions at the jet inlet.

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

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

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

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

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

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

Create an inlet with low-speed co-flow to make computations more stable.

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 **Co-Inlet**.

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

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

6 In the Ma_0 text field, type `Ma_co`.

Outlet 1

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

The outlet is placed far enough downstream so that subsonic conditions with atmospheric pressure can be applied.

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**.

Wall 2

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

The far-field boundary can be treated as a slip wall.

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

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

4 Locate the **Boundary Condition** section. From the **Wall condition** list, choose **Slip**.

Symmetry 1

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

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

3 From the **Selection** list, choose **Symmetry Boundary**.

A relatively coarse **Mesh 1** is used for ramping up the inlet pressure to a relatively high value.

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 **Extra coarse**.

Size 1

Right-click **Component 1 (comp1)** > **Mesh 1** and choose **Size**.

Size

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

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

Size 1

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

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

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

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **No-Slip Wall, Fine Mesh**.

6 Locate the **Element Size** section. From the **Predefined** list, choose **Finer**.

Size 2

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.

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

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

4 From the **Selection** list, choose **Refinement Edge**.

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

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

7 Click the **Custom** button.

8 Locate the **Element Size Parameters** section.

9 Select the **Maximum element size** checkbox. In the associated text field, type `4`.

Size 3

1 Right-click **Mesh 1** 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 From the **Selection** list, choose **Nozzle+Linking+Refinement**.

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

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

7 Click the **Custom** button.

8 Locate the **Element Size Parameters** section.

9 Select the **Maximum element growth rate** checkbox. In the associated text field, type `1.05`.

Free Tetrahedral 1

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

Boundary Layers 1

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

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

3 In the **Split for angles greater than** text field, type `270`.

Boundary Layer Properties

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

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

3 From the **Selection** list, choose **No-Slip Wall, Fine Mesh**.

4 Locate the **Layers** section. In the **Number of layers** text field, type 6.

Boundary Layer Properties 1

1 Right-click **Boundary Layer Properties** and choose **Duplicate**.

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

3 From the **Selection** list, choose **No-Slip Wall, Coarse Mesh**.

4 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

Study 1 is used to ramp up the inlet total pressure to 10[atm] using an auxiliary sweep. This way, initial values for further computations on finer meshes are quickly obtained.

STUDY 1

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

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

3 Clear the **Generate default plots** checkbox.

Step 2: Stationary

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

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

3 Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p_in	1.01 1.1 1.25 1.5 2 3 range(4,2,10)	atm

Solution 1 (sol1)

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

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

Take only seven iterations in each step of the auxiliary sweep, since intermediate converged solutions on the coarse mesh are not of interest.

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

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

5 From the **Termination technique** list, choose **Iterations or tolerance**.

6 In the **Number of iterations** text field, type 7.

Set up a hybrid preconditioner with velocity and pressure in **Multigrid 1** and temperature in **Multigrid 2**, to ensure convergence for computations on many cores.

7 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** > **AMG, fluid flow variables (hmnf)** node, then click **Multigrid 1**.

8 In the **Settings** window for **Multigrid**, click to expand the **Hybridization** section.

9 From the **Use as** list, choose **Multi preconditioner**.

10 In the **Preconditioner variables** list, choose **Turbulent Dissipation Rate (comp1.ep)**, **Reciprocal Wall Distance (comp1.G)**, **Wall Temperature, Downside (comp1.hmnf.TWall_d)**, **Wall Temperature, Upside (comp1.hmnf.TWall_u)**, **Turbulent Kinetic Energy (comp1.k)**, and **Temperature (comp1.T)**.

11 Under **Preconditioner variables**, click **Delete**.

12 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** right-click **AMG, fluid flow variables (hmnf)** and choose **Multigrid**.

Multigrid 2 uses settings that are default for the turbulence variables.

13 In the **Settings** window for **Multigrid**, locate the **General** section.

14 From the **Solver** list, choose **Smoothed aggregation AMG**.

15 In the **Maximum number of DOFs at coarsest level** text field, type 50000.

16 Select the **Construct prolongators componentwise** checkbox.

17 Clear the **Prolongator smoothing** checkbox.

18 Locate the **Hybridization** section. In the **Preconditioner variables** list, choose **Turbulent Dissipation Rate (comp1.ep)**, **Reciprocal Wall Distance (comp1.G)**, **Turbulent Kinetic Energy (comp1.k)**, **Pressure (comp1.p)**, and **Velocity Field (comp1.u)**.

19 Under **Preconditioner variables**, click **Delete**.

20 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** > **AMG, fluid flow variables (hmnf)** > **Multigrid 2** > **Presmoother** node.

21 Right-click **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** > **AMG, fluid flow variables (hmnf)** > **Multigrid 2** > **Presmoother** and choose **SOR Line**.

22 In the **Settings** window for **SOR Line**, locate the **Main** section.

23 From the **Sweep type** list, choose **SSOR**.

24 In the **Number of iterations** text field, type 0.

25 In the **Relaxation factor** text field, type 0.7.

26 From the **Multivariable method** list, choose **Uncoupled**.

27 Locate the **Secondary** section. In the **Relaxation factor** text field, type 0.5.

28 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** > **AMG, fluid flow variables (hmnf)** > **Multigrid 2** > **Postsmoother** node.

29 Right-click **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** > **AMG, fluid flow variables (hmnf)** > **Multigrid 2** > **Postsmoother** and choose **SOR Line**.

30 In the **Settings** window for **SOR Line**, locate the **Main** section.

31 From the **Sweep type** list, choose **SSOR**.

32 In the **Number of iterations** text field, type 1.

33 In the **Relaxation factor** text field, type 0.7.

34 From the **Multivariable method** list, choose **Uncoupled**.

35 Locate the **Secondary** section. In the **Relaxation factor** text field, type 0.5.

36 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** > **AMG, fluid flow variables (hmnf)** > **Multigrid 2** > **Coarse Solver** node.

37 Right-click **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** > **AMG, fluid flow variables (hmnf)** > **Multigrid 2** > **Coarse Solver** and choose **Direct**.

38 In the **Settings** window for **Direct**, locate the **General** section.

39 From the **Solver** list, choose **PARDISO**.

40 In the **Pivoting perturbation** text field, type 1.0E-13.

41 In the **Study** toolbar, click = **Compute**.

Create an error indicator for adaptive mesh refinement.

DEFINITIONS

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
dilatation_error_expression	-1.5*h*hmnf.divu/hmnf.c_s*sqrt((1+hmnf.gamma)/(2+(hmnf.gamma-1)*hmnf.Ma^2))		
shock_error_indicator	max(dilatation_error_expression,0)		
modified_error_indicator	1.5*h*(max(-hmnf.divu,0)+hmnf.sr/10)/hmnf.c_s*sqrt((1+hmnf.gamma)/(2+(hmnf.gamma-1)*hmnf.Ma^2))		
account_strain_influence	adaptlevel<1		
custom_error_indicator	1.5*h*(max(-hmnf.divu,0)+account_strain_influence*hmnf.sr/10)/hmnf.c_s*sqrt((1+hmnf.gamma)/(2+(hmnf.gamma-1)*hmnf.Ma^2))		
Y_e	(y-h_tot)/D_e		

At the first refinement level shock regions and shear layers are refined. At the second refinement level only shock regions are refined. Use the `adaptlevel` variable to control this.

MESH 2

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

Mesh 2 is constructed so that the nozzle and its close proximity have sufficient resolution, and adaptive mesh refinement will be applied only on the **Refinement Domain**. Far-field regions are kept coarse.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Extra coarse**.

Size 1

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

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element growth rate** text field, type `1.1`.

Size 1

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

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

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

4 From the **Selection** list, choose **Nozzle 1**.

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

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

7 Click the **Custom** button.

8 Locate the **Element Size Parameters** section.

9 Select the **Maximum element size** checkbox. In the associated text field, type `0.8`.

10 Select the **Minimum element size** checkbox. In the associated text field, type `0.1`.

Size 2

1 In the **Model Builder** window, 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 From the **Selection** list, choose **Nozzle 2**.

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

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

7 Click the **Custom** button.

8 Locate the **Element Size Parameters** section.

9 Select the **Maximum element size** checkbox. In the associated text field, type `0.3`.

10 Select the **Minimum element size** checkbox. In the associated text field, type `0.06`.

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 **Edge**.

4 From the **Selection** list, choose **Refinement Edge**.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.

6 Click the **Custom** button.

7 Locate the **Element Size Parameters** section.

8 Select the **Maximum element size** checkbox. In the associated text field, type `1`.

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 From the **Selection** list, choose **Refinement**.

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

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

7 Click the **Custom** button.

8 Locate the **Element Size Parameters** section.

9 Select the **Maximum element size** checkbox. In the associated text field, type `5`.

Free Tetrahedral 1

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

Boundary Layers 1

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

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

3 In the **Split for angles greater than** text field, type `270`.

Boundary Layer Properties

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

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

3 From the **Selection** list, choose **No-Slip Wall, Fine Mesh**.

4 Locate the **Layers** section. In the **Number of layers** text field, type `9`.

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 **Boundary Selection** section.

3 From the **Selection** list, choose **No-Slip Wall, Coarse Mesh**.

4 Locate the **Layers** section. In the **Number of layers** text field, type 6.

5 In the **Model Builder** window, right-click **Mesh 2** and choose **Build All**.

ROOT

Study 2 takes initial values from the last step of **Study 1**. It starts on the finer **Mesh 2** and improves it in two refinement steps to properly capture shear layers and shocks. This is needed because the exact shock structure is not known in advance, thus a reasonable (not excessively large) mesh cannot be built immediately.

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 the **Add Study** button in the window toolbar.

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

STUDY 2

Step 1: Wall Distance Initialization

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

2 Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

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

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

5 From the **Parameter value (p_in (atm))** list, choose **Last**.

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

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

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

9 From the **Parameter value (p_in (atm))** list, choose **Last**.

Two levels of adaptive mesh refinement using **Error indicator** are applied.

Step 2: Stationary

1 In the **Model Builder** window, click **Step 2: 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 From the **Error estimate** list, choose **Error indicator**.

5 Click **Add**.

6 In the table, enter the following settings:

Error expression	Active
custom_error_indicator	√

7 Find the **Mesh adaptation** subsection.

8 Select the **Maximum number of adaptations** checkbox. In the associated text field, type 2.

9 Locate the **Geometric Entity Selection for Adaptation** section. From the **Geometric entity level** list, choose **Domain**.

10 From the **Selection** list, choose **Refinement**.

11 In the **Model Builder** window, click **Study 2**.

12 In the **Settings** window for **Study**, locate the **Study Settings** section.

13 Select the **Store complete solver history** checkbox.

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.

3 In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 2** node, then click **Adaptive Mesh Refinement**.

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

5 Clear the **Allow coarsening** checkbox.

6 Find the **Mesh adaptation** subsection. In the **Maximum number of elements** text field, type 20000000.

Adjust settings in **Segregated 1** to achieve fast and smooth convergence. Notice that increasing **Initial CFL number** is not recommended in the general case.

7 In the **Model Builder** window, under **Study 2** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 2** click **Segregated 1**.

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

9 From the **Termination technique** list, choose **Iterations or tolerance**.

10 In the **Number of iterations** text field, type `120`.

11 In the **Initial CFL number** text field, type `25`.

12 In the **PID controller - proportional** text field, type `0.25`.

13 In the **PID controller - integral** text field, type `0.15`.

14 In the **PID controller - derivative** text field, type `0.15`.

15 In the **Target error estimate** text field, type `0.1`.

16 Clear the **Adaptive target error estimate** checkbox.

17 In the **CFL threshold** text field, type `9900`.

Set up a hybrid preconditioner with velocity and pressure in **Multigrid 1** and temperature in **Multigrid 2**, to ensure convergence for computations on many cores.

18 In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 2** > **AMG, fluid flow variables (hmnf)** node, then click **Multigrid 1**.

19 In the **Settings** window for **Multigrid**, locate the **Hybridization** section.

20 From the **Use as** list, choose **Multi preconditioner**.

21 In the **Preconditioner variables** list, choose **Turbulent Dissipation Rate (comp1.ep)**, **Reciprocal Wall Distance (comp1.G)**, **Wall Temperature, Downside (comp1.hmnf.TWall_d)**, **Wall Temperature, Upside (comp1.hmnf.TWall_u)**, **Turbulent Kinetic Energy (comp1.k)**, and **Temperature (comp1.T)**.

22 Under **Preconditioner variables**, click **Delete**.

23 In the **Model Builder** window, under **Study 2** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 2** right-click **AMG, fluid flow variables (hmnf)** and choose **Multigrid**.

Multigrid 2 uses settings that are default for the turbulence variables.

24 In the **Settings** window for **Multigrid**, locate the **General** section.

25 From the **Solver** list, choose **Smoothed aggregation AMG**.

26 In the **Maximum number of DOFs at coarsest level** text field, type `50000`.

27 Select the **Construct prolongators componentwise** checkbox.

28 Clear the **Prolongator smoothing** checkbox.

29 Locate the **Hybridization** section. In the **Preconditioner variables** list, choose **Turbulent Dissipation Rate (comp1.ep)**, **Reciprocal Wall Distance (comp1.G)**, **Turbulent Kinetic Energy (comp1.k)**, **Pressure (comp1.p)**, and **Velocity Field (comp1.u)**.

30 Under **Preconditioner variables**, click **Delete**.

31 In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 2** > **AMG, fluid flow variables (hmnf)** > **Multigrid 2** > **Presmoother** node.

32 Right-click **Study 2** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 2** > **AMG, fluid flow variables (hmnf)** > **Multigrid 2** > **Presmoother** and choose **SOR Line**.

33 In the **Settings** window for **SOR Line**, locate the **Main** section.

34 From the **Sweep type** list, choose **SSOR**.

35 In the **Number of iterations** text field, type `0`.

36 In the **Relaxation factor** text field, type `0.7`.

37 From the **Multivariable method** list, choose **Uncoupled**.

38 Locate the **Secondary** section. In the **Relaxation factor** text field, type `0.5`.

39 In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 2** > **AMG, fluid flow variables (hmnf)** > **Multigrid 2** > **Postsmoother** node.

40 Right-click **Study 2** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 2** > **AMG, fluid flow variables (hmnf)** > **Multigrid 2** > **Postsmoother** and choose **SOR Line**.

41 In the **Settings** window for **SOR Line**, locate the **Main** section.

42 From the **Sweep type** list, choose **SSOR**.

43 In the **Number of iterations** text field, type `1`.

44 In the **Relaxation factor** text field, type `0.7`.

45 From the **Multivariable method** list, choose **Uncoupled**.

46 Locate the **Secondary** section. In the **Relaxation factor** text field, type `0.5`.

47 In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 2** > **AMG, fluid flow variables (hmnf)** > **Multigrid 2** > **Coarse Solver** node.

48 Right-click **Study 2** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 2** > **AMG, fluid flow variables (hmnf)** > **Multigrid 2** > **Coarse Solver** and choose **Direct**.

49 In the **Settings** window for **Direct**, locate the **General** section.

50 From the **Solver** list, choose **PARDISO**.

51 In the **Pivoting perturbation** text field, type `1.0E-13`.

52 In the **Model Builder** window, click **Study 2**.

53 In the **Settings** window for **Study**, locate the **Study Settings** section.

54 Clear the **Generate default plots** checkbox.

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

RESULTS

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

2 In the **Settings** window for **Results**, locate the **Update of Results** section.

3 Select the **Only plot when requested** checkbox.

Define parameters for creating bounding boxes for plots.

Parameters

1 In the **Results** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
L1	0.05[m]	0.05 m	
L2	6*h_tot	0.54 m	
L3	2.3*L1	0.115 m	
L4	0.04[m]	0.04 m	
L5	3.5*h_tot	0.315 m	

Create datasets for 3D and 2D plot groups. Start with the isosurface for the supersonic jet envelope.

Isosurface 1

1 In the **Results** toolbar, click **More Datasets** and choose **Isosurface**.

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

3 From the **Dataset** list, choose **Study 2/Adaptive Mesh Refinement Solutions 1 (sol5)**.

4 Locate the **Expression** section. In the **Expression** text field, type `hmnf.Ma`.

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

6 In the **Levels** text field, type `1`.

Isosurface 2

1 Right-click **Isosurface 1** and choose **Duplicate**.

Specify isosurfaces for a range of Mach numbers.

2 In the **Settings** window for **Isosurface**, locate the **Levels** section.

3 In the **Levels** text field, type `range(2.0,0.125,3.5)`.

Complement **Isosurface 1** using rotation and reflection.

Sector 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Sector 3D**.

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

3 From the **Dataset** list, choose **Isosurface 1**.

4 Locate the **Axis Data** section. In row **Point 1**, set **y** to 1.

5 In row **Point 2**, set **z** to 0.

6 Locate the **Symmetry** section. In the **Number of sectors** text field, type 4.

7 From the **Transformation** list, choose **Rotation and reflection**.

Complement **Isosurface 2** using rotation and reflection.

Sector 3D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Sector 3D**.

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

3 From the **Dataset** list, choose **Isosurface 2**.

4 Locate the **Symmetry** section. From the **Sectors to include** list, choose **Manual**.

5 In the **Start sector** text field, type 1.

6 In the **Number of sectors to include** text field, type 3.

7 Locate the **Axis Data** section. In row **Point 1**, set **y** to 1.

8 In row **Point 2**, set **z** to 0.

9 Locate the **Symmetry** section. In the **Number of sectors** text field, type 4.

10 From the **Transformation** list, choose **Rotation and reflection**.

Create a minor axis (vertical plane) surface.

Surface 1

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

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

3 From the **Dataset** list, choose **Study 2/Adaptive Mesh Refinement Solutions 1 (sol5)**.

4 Locate the **Parameterization** section. From the **x- and y-axes** list, choose **yx-plane**.

5 Locate the **Selection** section. From the **Selection** list, choose **Vertical Plane**.

Surface 2

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

Create a major axis (horizontal plane) surface.

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

3 From the **x- and y-axes** list, choose **yz-plane**.

4 Locate the **Selection** section. From the **Selection** list, choose **Horizontal Plane**.

Surface 3

1 In the **Model Builder** window, under **Results** > **Datasets** right-click **Surface 1** and choose **Duplicate**.

Create a dummy (vertical) surface for creating a cut line to be used as a geometric entity in a 2D plot.

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

3 From the **Dataset** list, choose **Study 1/Solution Store 1 (sol2)**.

4 Locate the **Selection** section. From the **Selection** list, choose **Vertical Extended Plane**.

Cut Line 2D 1

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 **Surface 3**.

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

5 In row **Point 2**, set **x** to `L2`.

6 Select the **Additional parallel lines** checkbox.

7 In the **Distances** text field, type `range(L3,L3,2*L3)`.

Export high-resolution images of the meshes.

Image 1

1 In the **Results** toolbar, click **Image**.

2 In the **Settings** window for **Image**, choose **Presentation and document** from the **Preset** list.

3 Locate the **Image** section. In the **Width** text field, type `3165`.

4 In the **Height** text field, type `2374`.

5 In the **Resolution** text field, type `90`.

6 Locate the **Scene** section. In the tree, select **Model (root)** > **Component 1 (comp1)** > **Meshes** > **Mesh 1**.

7 Click **Use as Source**.

8 Locate the **Output** section. From the **Target** list, choose **File**.

9 Locate the **Scene** section. From the **View** list, choose **View 2**.

10 Locate the **Output** section. In the **Filename** text field, type `rectangular_nozzle_mesh_1_HiRes.png`.

11 Click **Export**.

Image 2

1 Right-click **Image 1** and choose **Duplicate**.

2 In the **Settings** window for **Image**, locate the **Scene** section.

3 In the tree, select **Model (root)** > **Component 1 (comp1)** > **Meshes** > **Mesh 2**.

4 Click **Use as Source**.

5 From the **View** list, choose **View 3**.

6 Locate the **Output** section. In the **Filename** text field, type `rectangular_nozzle_mesh_2_HiRes.png`.

7 Click **Export**.

Image 3

1 Right-click **Image 2** and choose **Duplicate**.

2 In the **Settings** window for **Image**, locate the **Scene** section.

3 From the **View** list, choose **View 4**.

4 Locate the **Output** section. In the **Filename** text field, type `rectangular_nozzle_mesh_3_HiRes.png`.

5 Click **Export**.

Image 4

1 Right-click **Image 3** and choose **Duplicate**.

2 In the **Settings** window for **Image**, locate the **Scene** section.

3 In the tree, select **Model (root)** > **Component 1 (comp1)** > **Meshes** > **Level 1 Adapted Mesh 1**.

4 Click **Use as Source**.

5 Locate the **Output** section. In the **Filename** text field, type `rectangular_nozzle_mesh_4_HiRes.png`.

6 Click **Export**.

Image 5

1 Right-click **Image 4** and choose **Duplicate**.

2 In the **Settings** window for **Image**, locate the **Scene** section.

3 In the tree, select **Model (root)** > **Component 1 (comp1)** > **Meshes** > **Level 2 Adapted Mesh 2**.

4 Click **Use as Source**.

5 Locate the **Output** section. In the **Filename** text field, type `rectangular_nozzle_mesh_5_HiRes.png`.

6 Click **Export**.

Centerline Mach Number

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

2 In the **Settings** window for **1D Plot Group**, type `Centerline Mach Number` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Adaptive Mesh Refinement Solutions 1 (sol5)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `Y_e`.

7 Select the **y-axis label** checkbox. In the associated text field, type `Mach number`.

Line Graph 1

1 Right-click **Centerline Mach Number** and choose **Line Graph**.

2 Select Edges 11, 16, 21, and 34 only.

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

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

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

6 In the **Expression** text field, type `Y_e`.

7 Click to expand the **Legends** section. Select the **Show legends** checkbox.

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

9 In the table, enter the following settings:

Legends
`No refinement`
`Refinement 1`
`Refinement 2`

10 In the **Centerline Mach Number** toolbar, click **Plot**.

Centerline Velocity Divergence

1 In the **Model Builder** window, right-click **Centerline Mach Number** and choose **Duplicate**.

2 In the **Model Builder** window, click **Centerline Mach Number 1**.

3 In the **Settings** window for **1D Plot Group**, type `Centerline Velocity Divergence` in the **Label** text field.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Decimal logarithm of velocity divergence`.

5 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Line Graph 1

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

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

3 In the **Expression** text field, type `sign(hmnf.divu)*log10(abs(hmnf.divu))`.

4 In the **Centerline Velocity Divergence** toolbar, click **Plot**.

Centerline Pressure

1 In the **Model Builder** window, right-click **Centerline Velocity Divergence** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Centerline Pressure` in the **Label** text field.

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type `p [atm]`.

Line Graph 1

1 In the **Model Builder** window, expand the **Centerline Pressure** node, then click **Line Graph 1**.

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

3 In the **Expression** text field, type `p/(1[atm])`.

4 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.

5 In the **Centerline Pressure** toolbar, click **Plot**.

Jet Vertical-Horizontal Asymmetry

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Jet Vertical-Horizontal Asymmetry` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Top: Axial velocity (m/s) Middle:  Cross-stream velocity (m/s) Bottom: Logarithm of velocity divergence`.

5 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Adaptive Mesh Refinement Solutions 1 (sol5)**.

6 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

7 In the **Jet Vertical-Horizontal Asymmetry** toolbar, click **Plot**.

8 Clear the **Plot dataset edges** checkbox.

9 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

10 Clear the **Show legends** checkbox.

Axial Velocity

1 Right-click **Jet Vertical-Horizontal Asymmetry** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Axial Velocity` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `v`.

Selection 1

1 Right-click **Axial Velocity** and choose **Selection**.

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

3 From the **Selection** list, choose **Vertical+Horizontal Planes**.

Filter 1

1 In the **Model Builder** window, right-click **Axial Velocity** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `(x<L4)*(z<L4)*(y<L5)`.

Cross-Stream Velocity

1 Right-click **Axial Velocity** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Cross-Stream Velocity` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `sqrt(u^2+w^2)`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

Transformation 1

1 Right-click **Cross-Stream Velocity** and choose **Transformation**.

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

3 In the **x** text field, type `-1.25*L4`.

4 In the **z** text field, type `L4`.

Compression-Expansion Strength

1 In the **Model Builder** window, right-click **Cross-Stream Velocity** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Compression-Expansion Strength` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `sign(hmnf.divu)*log10(abs(hmnf.divu))`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayScale**.

Transformation 1

1 In the **Model Builder** window, expand the **Compression-Expansion Strength** node, then click **Transformation 1**.

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

3 In the **x** text field, type `-2*1.25*L4`.

4 In the **z** text field, type `2*L4`.

5 In the **Jet Vertical-Horizontal Asymmetry** toolbar, click **Plot**.

6 Adjust the camera to reproduce **Figure 6**.

Show legends on the figure and zoom extents.

Jet Vertical-Horizontal Asymmetry

1 In the **Model Builder** window, under **Results** click **Jet Vertical-Horizontal Asymmetry**.

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

3 Select the **Show legends** checkbox.

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

Supersonic Jet Envelope

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Supersonic Jet Envelope` in the **Label** text field.

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

4 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

5 In the **Supersonic Jet Envelope** toolbar, click **Plot**.

6 Locate the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Turbulence Reynolds number at the surface Ma=1`.

8 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

9 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

Surface 1

1 Right-click **Supersonic Jet Envelope** and choose **Surface**.

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

3 In the **Expression** text field, type `hmnf.muT/hmnf.mu`.

4 In the **Supersonic Jet Envelope** toolbar, click **Plot**.

Shock Diamonds Isosurfaces

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Shock Diamonds Isosurfaces` in the **Label** text field.

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

4 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

5 In the **Shock Diamonds Isosurfaces** toolbar, click **Plot**.

6 Locate the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Isosurfaces: Mach number`.

8 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

9 Locate the **Color Legend** section. From the **Position** list, choose **Bottom**.

10 Select the **Show maximum and minimum values** checkbox.

Surface 1

1 Right-click **Shock Diamonds Isosurfaces** and choose **Surface**.

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

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

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

Turbulence Reynolds Number

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Turbulence Reynolds Number` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Adaptive Mesh Refinement Solutions 1 (sol5)**.

4 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

5 In the **Turbulence Reynolds Number** toolbar, click **Plot**.

6 Locate the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Turbulence Reynolds number`.

8 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

9 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Surface 1

1 Right-click **Turbulence Reynolds Number** and choose **Surface**.

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

3 In the **Expression** text field, type `hmnf.muT/hmnf.mu`.

Selection 1

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

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

3 From the **Selection** list, choose **Vertical+Horizontal Planes**.

4 Select Boundaries 7, 9, 11, 13, 15, 17, 23, 25, and 28 only.

Turbulence Reynolds Number

1 In the **Model Builder** window, under **Results** click **Turbulence Reynolds Number**.

2 In the **Turbulence Reynolds Number** toolbar, click **Plot**.

Streamlines

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Streamlines` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Adaptive Mesh Refinement Solutions 1 (sol5)**.

4 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

5 Locate the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Jet and entrainment streamlines, vertical (Top) and horizontal (Bottom) planes`.

7 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

8 Locate the **Color Legend** section. Clear the **Show legends** checkbox.

Vertical Jet Streamlines

1 In the **Streamlines** toolbar, click **More Plots** and choose **Streamline Surface**.

2 In the **Settings** window for **Streamline Surface**, type `Vertical Jet Streamlines` in the **Label** text field.

3. Locate the **Surface Selection** section. From the **Selection** list, choose **Vertical Extended Plane**.
4. Locate the **Streamline Positioning** section. From the **Positioning** list, choose **On selected edges**.
5. In the **Number** text field, type `25`.
6. Locate the **Edge Selection** section. Click **Paste Selection**.
7. In the **Paste Selection** dialog, type `12` in the **Selection** text field.
8. Click **OK**.
9. In the **Settings** window for **Streamline Surface**, locate the **Coloring and Style** section.
10. Find the **Line style** subsection. From the **Type** list, choose **Tube**.
11. Select the **Radius scale factor** checkbox. In the associated text field, type `0.1`.
12. Find the **Point style** subsection. From the **Color** list, choose **Custom**.
13. On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.
14. Click **Define custom colors**.
15. Set the RGB values to 128, 0, and 0, respectively.
16. Click **Add to custom colors**.
17. Click **Show color palette only** or **OK** on the cross-platform desktop.

Vertical Entrainment Streamlines

1. Right-click **Vertical Jet Streamlines** and choose **Duplicate**.
2. In the **Settings** window for **Streamline Surface**, type `Vertical Entrainment Streamlines` in the **Label** text field.
3. Locate the **Edge Selection** section. In the list box, select **12**.
4. Locate the **Streamline Positioning** section. In the **Number** text field, type `75`.
5. Locate the **Edge Selection** section. Click **Paste Selection**.
6. In the **Paste Selection** dialog, type `102` in the **Selection** text field.
7. Click **OK**.
8. In the **Settings** window for **Streamline Surface**, locate the **Coloring and Style** section.
9. Click **Define custom colors**.
10. Set the RGB values to 0, 128, and 192, respectively.
11. Click **Add to custom colors**.
12. Click **Show color palette only** or **OK** on the cross-platform desktop.

Horizontal Jet Streamlines

1 In the **Model Builder** window, right-click **Vertical Jet Streamlines** and choose **Duplicate**.

2 In the **Settings** window for **Streamline Surface**, type `Horizontal Jet Streamlines` in the **Label** text field.

3 Locate the **Surface Selection** section. From the **Selection** list, choose **Horizontal Extended Plane**.

4 Locate the **Edge Selection** section. In the list box, select **12**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type `10` in the **Selection** text field.

7 Click **OK**.

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

9 Click **Define custom colors**.

10 Set the RGB values to 64, 0, and 0, respectively.

11 Click **Add to custom colors**.

12 Click **Show color palette only** or **OK** on the cross-platform desktop.

Transformation 1

1 Right-click **Horizontal Jet Streamlines** and choose **Transformation**.

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

3 Clear the **Move** checkbox.

4 Select the **Rotate** checkbox.

5 From the **Axis type** list, choose **Y-axis**.

6 In the **Angle** text field, type `-90`.

Horizontal Entrainment Streamlines

1 In the **Model Builder** window, right-click **Horizontal Jet Streamlines** and choose **Duplicate**.

2 In the **Settings** window for **Streamline Surface**, type `Horizontal Entrainment Streamlines` in the **Label** text field.

3 Locate the **Edge Selection** section. In the list box, select **10**.

4 Locate the **Streamline Positioning** section. In the **Number** text field, type `75`.

5 Locate the **Edge Selection** section. Click **Paste Selection**.

6 In the **Paste Selection** dialog, type `1` in the **Selection** text field.

7 Click **OK**.

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

9 Click **Define custom colors**.

10 Set the RGB values to 0, 64, and 64, respectively.

11 Click **Add to custom colors**.

12 Click **Show color palette only** or **OK** on the cross-platform desktop.

13 In the **Streamlines** toolbar, click **Plot**.

Jet Asymmetry Planar Perspective

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

2 In the **Settings** window for **2D Plot Group**, type `Jet Asymmetry Planar Perspective` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Top: Logarithm of vorticity Middle: Temperature (K) Bottom: Mach number`.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

6 Locate the **Color Legend** section. Clear the **Show legends** checkbox.

Ma Vertical Plane

1 Right-click **Jet Asymmetry Planar Perspective** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Ma Vertical Plane` in the **Label** text field.

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

Filter 1

1 Right-click **Ma Vertical Plane** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `(x<L1)*(z<L1)*(y<L2)`.

Transformation 1

In the **Model Builder** window, right-click **Ma Vertical Plane** and choose **Transformation**.

Ma Horizontal Plane

1 Right-click **Ma Vertical Plane** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Ma Horizontal Plane` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Surface 2**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Ma Vertical Plane**.

Transformation 1

1 In the **Model Builder** window, expand the **Ma Horizontal Plane** node, then click **Transformation 1**.

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

3 Select the **Scale** checkbox.

4 In the **y** text field, type `-1`.

T Vertical Plane

1 In the **Model Builder** window, right-click **Ma Vertical Plane** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `T Vertical Plane` in the **Label** text field.

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

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Thermal**.

Transformation 1

1 In the **Model Builder** window, expand the **T Vertical Plane** node, then click **Transformation 1**.

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

3 In the **y** text field, type `L3`.

T Horizontal Plane

1 In the **Model Builder** window, right-click **Ma Horizontal Plane** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `T Horizontal Plane` in the **Label** text field.

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

4 Locate the **Inherit Style** section. From the **Plot** list, choose **T Vertical Plane**.

Transformation 1

1 In the **Model Builder** window, expand the **T Horizontal Plane** node, then click **Transformation 1**.

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

3 In the **y** text field, type `L3`.

Decimal Logarithm Vorticity Vertical Plane

1 In the **Model Builder** window, right-click **T Vertical Plane** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Decimal Logarithm Vorticity Vertical Plane` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `log10(hmnf.vort_magn)`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

Transformation 1

1 In the **Model Builder** window, expand the **Decimal Logarithm Vorticity Vertical Plane** node, then click **Transformation 1**.

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

3 In the **y** text field, type `2*L3`.

Decimal Logarithm Vorticity Horizontal Plane

1 In the **Model Builder** window, right-click **T Horizontal Plane** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `Decimal Logarithm Vorticity Horizontal Plane` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `log10(hmnf.vort_magn)`.

4 Locate the **Inherit Style** section. From the **Plot** list, choose **Decimal Logarithm Vorticity Vertical Plane**.

Transformation 1

1 In the **Model Builder** window, expand the **Decimal Logarithm Vorticity Horizontal Plane** node, then click **Transformation 1**.

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

3 In the **y** text field, type `2*L3`.

Line 1

1 In the **Model Builder** window, right-click **Jet Asymmetry Planar Perspective** and choose **Line**.

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

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

4 Locate the **Expression** section. In the **Expression** text field, type `1`.

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

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

7 In the **Jet Asymmetry Planar Perspective** toolbar, click **Plot**.

8 Adjust camera to reproduce **Figure 7**.

Show legends on the figure.

Jet Asymmetry Planar Perspective

1 In the **Model Builder** window, click **Jet Asymmetry Planar Perspective**.

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

3 Select the **Show legends** checkbox.

Calculate maximum, minimum and average values.

Evaluation Group 1

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

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

3 From the **Dataset** list, choose **Study 2/Adaptive Mesh Refinement Solutions 1 (sol5)**.

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

Surface Maximum 1

1 Right-click **Evaluation Group 1** and choose **Maximum** > **Surface Maximum**.

2 Select Boundaries 23 and 25 only.

3 In the **Settings** window for **Surface Maximum**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
hmnf.Ma	1	Mach number
hmnf.U	m/s	Velocity magnitude
hmnf.muT/hmnf.mu	1	

Volume Minimum 1

1 In the **Model Builder** window, right-click **Evaluation Group 1** and choose **Minimum** > **Volume Minimum**.

2 In the **Settings** window for **Volume Minimum**, locate the **Selection** section.

3 From the **Selection** list, choose **All domains**.

Surface Average 1

1 Right-click **Evaluation Group 1** and choose **Average** > **Surface Average**.

2 Select Boundary 27 only.

3 In the **Settings** window for **Surface Average**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
u*hmnf.nxmesh+v*hmnf.nymesh+w*hmnf.nzmesh	m/s	

5 In the **Evaluation Group 1** toolbar, click **Evaluate**.