

subject: Advanced Computational Fluid Dynamics
Komputerowa Analiza Przepływów

Exercise 4

Projectile moving through a barrel and out of the muzzle

Purpose

The purpose of this exercise is to simulate an expansion of gases moving out of the muzzle. Some insight into the physical processes taking place inside the silencer can be gained (expansion and compression of the pressure waves).

Duration: 2x2h

Mesh

The computational mesh is shown in Fig.1. The circle radius in the front part of the projectile is $R=4.5\text{mm}$. The projectile is an empty zone. No mesh should be created to simulate the projectile itself. The length of the face A should be somewhat larger than the length of the face C. The other dimensions (length of the projectile, size of the silencer) are arbitrary. The interface_outer is attached to the face C. The interface_inner is attached to the zones A and B together. As shown, the interface_inner consists of two line segments, one in front and the other behind the projectile.

The line segments h1,h2,h3,v1,v2 and v3 (infinitesimally thin walls) correspond to the silencer. They can be created using the **Tools/Project** option in Workbench Design Modeler. The steps are as follows:

- Sketch the face C (xy-plane, rectangle) and create the surface **Concept/Surfaces from sketches**
- Open new sketch (xy-plane) and draw six line segments h1,h2,h3,v1,v2 and v3 (dimension is arbitrary but the vertical lines should not be linked to the bottom side of face C).
- Switch to **Modeling** mode. Create lines **Concept/Lines from Sketches**.
- Project the lines onto the face C **Tools/Project** (Target is the face C).
- Specify the names of the line segments h1,h2,h3,v1,v2 and v3 . Select given line, right click and select **Name Selection**

Make an unstructured mesh in front of the projectile. The mapped mesh scheme might be difficult to apply there due to narrow fluid zone in vicinity of the projectile.

The boundary conditions and zones should be defined as shown in Fig. 1.

Fluent

The exercise will consist of two stages. In the first stage, the flow will be simulated in the computational domain as shown in Fig. 1 (silencer will consist of six line segments h_1, h_2, h_3, v_1, v_2 and v_3). In the second stage, the horizontal walls will be removed. In Fluent, the walls h_1, h_2 and h_3 should be simply redefined to interior lines.

It is assumed that the projectile is moving at constant speed of 300 m/s. The moving meshes should be defined for the faces A and B (translational velocity). Note that for B zone the fluid velocity should be set to $U = -300 \text{ m/s}$ (minus), so the initial absolute air velocity will be zero there.

The steps are as follows:

1. Read the mesh. **File/Read/Mesh**. Scale the mesh in **Mesh/Scale**, if needed. The mesh was created in mm. Press the **Scale** button, to confirm, after selecting mm from the list of options in **Mesh Was Created In**.
2. **Define/General**. The solver settings are: Type: Density Based, Time: Transient, 2D-Space: Axisymmetric.
3. In **Define/Units** change the unit for pressure from Pa to atmosphere. Press the **New** button after selecting the pressure and atm.
4. In **Define/Mesh Interfaces/Create Edit** set-up the interface between moving (A and B) and stationary (C) fluid zones. All the interfaces/interface with the name 'interface_inner' should be selected in the one window and all the remaining interfaces/interface with the name 'interface_outer' should be selected in the other window. Provide an arbitrarily name to the interface under **Mesh Interface**.
5. Check the quality of the mesh in **Mesh/Check**. If the negative volumes appear, the mesh should be repaired in mesher (mesh coarsening in some regions might be helpful in this case).
6. Activate the Spalart-Allmaras turbulence model in **Define/Models/Viscous**.
7. The gas behind the projectile (zone A) and the air in the zones B and C are treated as compressible air. Select the ideal-gas for a fluid density in **Define/Materials/Fluid**.
8. Specify the first-order upwind discretization scheme for the convective terms in the momentum and transport equations. Set the Courant number to unity.
9. Define the moving mesh for the zones A and B in **Define/Cell Zone Conditions/Edit**. Translational velocity should be set to 300m/s.
10. Specify the following boundary conditions:
 - pressure inlet1: Set the Gauge Total Pressure and Supersonic/Initial Gauge Pressure to 200-500 atm and the turbulent viscosity ratio to 1. Temperature: 1000K.
 - pressure outlet1: Set the turbulent viscosity ratio to 0.1.
 - pressure outlet2: Set the turbulent viscosity ratio to 0.1.
11. Initialize the solution in the **Solution/Initialization**. The default values for pressure and velocity should be used (zero). The turbulent viscosity should be set to $0.01 \text{ m}^2/\text{s}$. Next, the **Patch** option should be used to initialize the solution in the zones A and B as shown in Fig. 1. The settings for the zone A should be the same as for the pressure inlet1 (p. no. 9).

12. One can verify the initial pressure, velocity and temperature fields in **Display/Graphics and Animations/Contours/Set up**. Use **Display/View/Front/Apply** to see the overall computational domain.
13. Save the case and data file in your working directory, **File/Write Case Data**.
14. Set-up the animation sequences in **Solve/Calculation Activities** under **Solution Animation**.
 - Visualize the pressure contours. Under **Animation Sequences** specify 2. Under **When** select the Time Step. In Define set the Window to 3 and press the Set button. In the **Display Type** take Contours of Static Pressure. Deactivate the **Auto Range** under Options and set 0 and 5 atm for min and max pressure, respectively. Click **Ok** button to confirm.
 - Visualize the velocity magnitude (either contour or the velocity vectors can be specified). Under **Animation Sequences** specify 2. Under **When** select the Time Step. In Define set the Window to 4 and press the Set button. In the **Display Type** take the Contours of Velocity Magnitude or the Velocity Vectors. Deactivate the **Auto Range** and set the appropriate values for min and max velocity. Click **Ok** button.
 - Visualize the temperature. Under **Animation Sequences** specify 2. Under **When** select the Time Step. In Define set the Window to 5 and press the Set button. In the **Display Type** take the Contours of Static Temperature. Click **Ok** button.
15. Monitor the pressure force acting on one of the vertical walls v1,v2 or v3, **Solve/Monitors/Surface Monitors**. Select the v1,v2 or v3 under **Surfaces**. Select the integral of static pressure. Activate the write and plot options. Set the window number to 10. Use the Flow Time and Time step in the **X Axis** and **Get data Every**, respectively.
16. In **Solve/Monitors/Surface Monitors**, monitor the static pressure in one point located in the zone C (relatively far from the silencer). Select **Area-Weighted Average** under **Report Type**. Set the window number to 11. In order to create the point, switch to **New Surface**, then to **Point**. Create the point with the mouse button. Select the created point under **Surfaces**. The other settings as above.
17. In **Solve/Run Calculation** set the time step to 1e-8s and the number of time steps to 10. Activate the calculations. Next, increase the time step to 1e-7 and rise the number of time steps to 100. Increase the number of time steps and complete the calculation.

To finish the first part of the exercise the following animations/pictures have to be presented

1. Animations of pressure, velocity contours/vectors and temperature.
2. Monitoring of the time-varying pressure force acting on one of the walls v1,v2 or v3 and the pressure fluctuation in a one point located in the zone C (outside of silencer)

In order to finish the second part of the exercise the horizontal line segments h1,h2 and h3 should be redefined to the interior lines (in Fluent, boundary conditions). The computations should be initialized and the animations and force monitoring should be set-up once again, as discussed above.

The animations/pictures and monitoring of forces obtained in the second part of the exercise have to be compared with the animations/pictures obtained before.

What is the pressure force acting on the walls of silencer ? Are the forces large ?

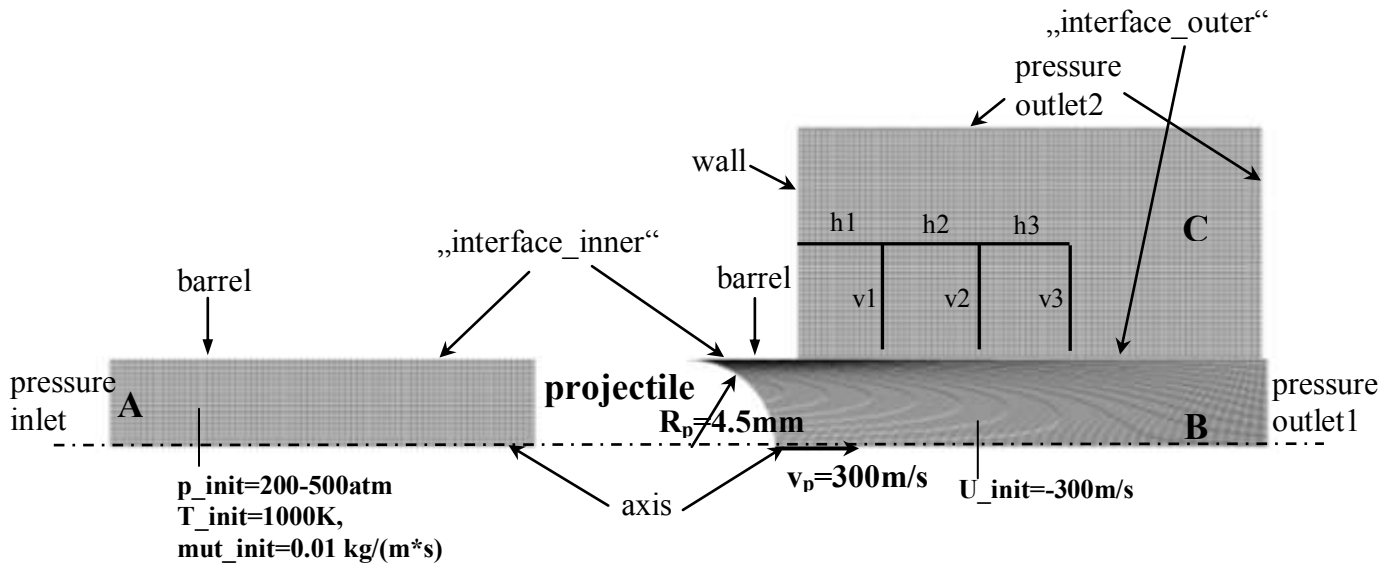


Fig 1. Computational mesh, boundary and initial conditions. The “interface_inner” consist of two line segments (one in front and the other behind the projectile, top side of zone B). The “interface_outer” corresponds to bottom side of rectangle C. The three fluid zones have to be created: A,B and C. The line segments h_1, h_2, h_3, v_1, v_2 and v_3 represent infinitesimally thin walls of the silencer. p_{init} , T_{init} and $\mu_{t\text{init}}$ denote the initial pressure, temperature and the turbulent viscosity, respectively. The length of domains A,B and C (and the height of domain C) should not be less than 10 projectile lengths (here the model is out of scale).