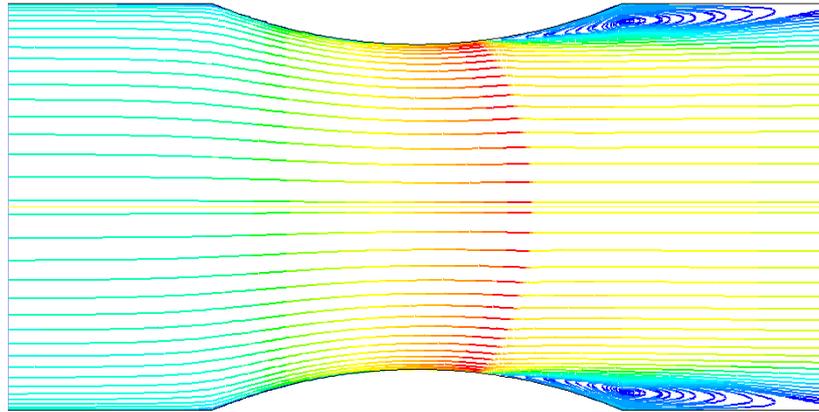


# Exercise 5: Steady axisymmetric flow through a converging-diverging nozzle

---



## *Purpose*

Gain knowledge on physics of compressible flows. Get familiar with simulations using a coupled density-based solver and application of grid-adaptation technique to improve the resolution at shock waves.

## *Summary*

In this tutorial, FLUENT's density-based implicit solver is used to predict the flow through a nozzle assuming different boundary conditions. Due to axis-symmetry, only half of the geometry shown in Fig. 0.1 is used in simulation.

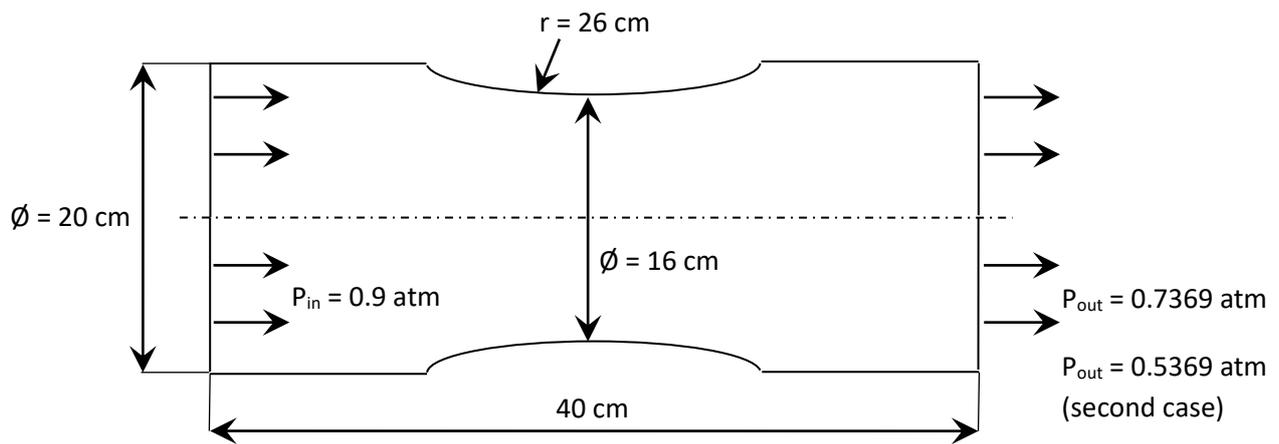


Fig 0.1: Sketch of geometry with assigned values of pressure the inlet and outlet. The domain is copied once along the symmetry axis.

### Creating geometry

First you must prepare surface, using DesignModeler. Because this case is axis symmetric, only top half of the geometry needs to be drawn. It is very important to place axis of the nozzle along X axis of the coordinate system (In Fluent, by default, X axis is automatically set as model's axis of symmetry).

To draw geometry you only need to use five straight lines and an arc, then add appropriate constraints and dimensions. When finished, create surface from your sketch. In properties of body that you created select that it should be treated as fluid.

### Creating mesh

We want to have 120 elements along axis. One way to fulfill this condition, is to define size of elements on entire surface (calculate size of element yourself – 120 divisions need to be on a 400 mm line).

Now stretch the grid towards the wall (fig. 0.2) by using either Inflation or Bias function on outlet and inlet edges. When you insert sizing of an edge, at the bottom you have bias options. Select appropriate bias type and insert bias factor equal to 11. To have a denser grid along the whole wall, you now need to add Mapped face mesh.

Because we strictly define mesh by ourselves it is better to turn off Advanced Size Function in Mesh sizing options.

Specify the names of edges as: “pressure\_inlet” (left vertical line segment), “pressure\_outlet” (right vertical line segment) and “axis” (bottom). Export mesh (as \*.msh file) so you could open it in stand alone Fluent.

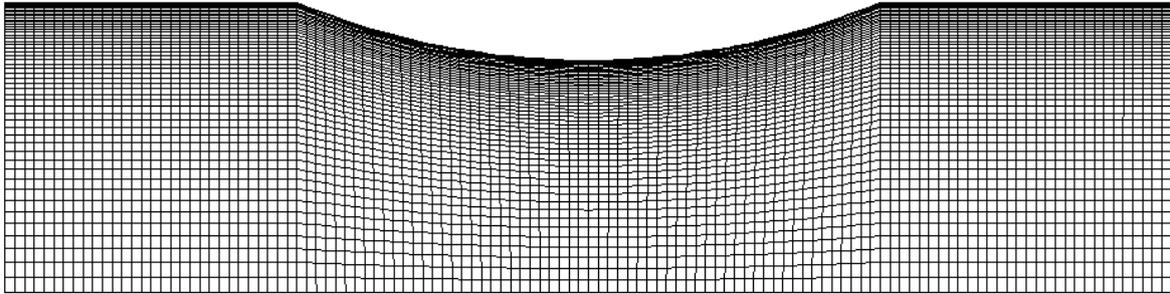


Fig 0.2: Mesh view

### *Solver settings and calculations*

1. For calculation we use only top half of the nozzle, but mesh and results can be reflected for displaying the whole nozzle using (**Results/Graphics/Views**)
2. It is convenient to change pressure unit to atmospheres (**General/Units**)
3. Set the density based solver (coupled solver). This solver is suggested for flows at high Mach number (shock waves may appear so the system of equations has to be solved in coupled fashion). For flows at low Mach number the pressure based (segregated) solver can be used. Set the Courant-Friedrichs-Lewy (CFL) number to 5. **Solution/Controls**

*There are two solvers in Fluent: the pressure- and density-based. In pressure-based solver the fulfillment of the continuity equation is obtained by solution of the Poisson equation for pressure. The velocity and pressure is corrected by inclusion of appropriate correction term (resulting from solution of the Poisson equation). For compressible flows the density is obtained from equation of state. For incompressible flows the density can be taken as a constant (liquids) or it can be computed as function of the temperature and operating pressure (gases). In density-based solver, in contrast to the pressure-based solver, the density is obtained by solution of the coupled system of equation consists of momentum, continuity and energy (where appropriate). The pressure is obtained from the equation of state.*

*By default the pressure-based solver is a segregated solver. It means that the momentum, pressure-correction (continuity) and all transport equations (energy, turbulence model) are solved sequentially one after another. Because the governing equations are nonlinear and coupled the solution loop must be carried out to obtain converged solution. In recent versions of Fluent the pressure-based solver can be also used as coupled solver. The coupled solver means that velocity, pressure and temperature (where appropriate) fields (density-based solver) or velocity and pressure-correction fields (pressure-based solver) are obtained at once by solution of large (coupled) system of equations. The other transport equations (e.g. turbulence model, species concentration) are solved sequentially.*

4. Turn on Energy equation – in compressible flow, temperature changes can't be omitted.
5. Choose k- $\omega$  SST turbulence model.
6. Change the material properties (set ideal gas for density) – air cannot be treated as constant density medium.
7. Boundary conditions
  - a. Set operating pressure to 0 atm (bear in mind that now all pressure's values set in boundary conditions will be equal to absolute pressure)
  - b. Inlet (pressure inlet – set the gauge total (or stagnation) pressure of the inflow stream)
    - i. Gauge Total Pressure  $p_0=0.9$  [atm]
    - ii. Supersonic/Initial Gauge Pressure  $p_{\text{static}}=0.7369$  [atm] (*this sets the static pressure on the boundary when the flow becomes (locally) supersonic. It is also used to compute the initial value of the static pressure if the flow is subsonic.*)
    - iii. Set the turbulent intensity to 1.5% and viscosity ratio to 10
  - c. Outflow (pressure outlet –sets the gauge pressure at the outflow boundary)
    - i. gauge pressure  $p_{\text{static}}=0.7369$  [atm]
    - ii. backflow turbulent intensity 1.5, turbulent to molecular viscosity ratio 10
8. Solution methods – set to 1st order scheme for all equations.
9. Set the initial value of the Courant number to 5 (**Solution/Controls**). While writing your report specify for which value of the Courant number the solution has been obtained.
 

*The CFL number ( $CFL=U\Delta t/\Delta x$  where  $U$  is the velocity,  $\Delta t$  is the time step,  $\Delta x$  is the mesh cell size) is a solution parameter which allows to control the convergence of the iterative process. This parameter is used in coupled solvers. Each set of equations is solved in time-dependent fashion (both steady and unsteady problems are solved in pseudo-time step). The Runge-Kutta scheme is employed for time-matching within each iteration step for steady problems.*

*The CFL number can be reduced to 1-5 in order to avoid a convergence problems (e.g. at beginning of simulation). If the convergence is good it can be raised to high values 10-50.*
10. In **Solution/Controls**, set Limits: minimum temperature to 200 K and maximum temperature to 400K (*In order to keep the solution stable under extreme conditions, Fluent provides limits that keep the solution within an acceptable range*)

11. In order to control convergence, monitor difference between inlet and outlet mass flow rate. First in Residuals monitors, disable Convergence criterion (set it to none). Create a new Surface Monitor in **Solution/Monitors/Report Plots** used for reporting Mass flow rate at the inlet and outlet (select them both under Surfaces – Fluent will automatically calculate the difference), remember to tick Plot under Options to see graph during calculations.
12. Set graphic window division into two parts (**View / Graphics Window Layout**)
13. Initialize from inlet and start calculations, stop them when mass flow rate stops changing.
14. To check the exact value of inlet and outlet mass flow difference go to **Results/Reports/Fluxes** and select appropriate surfaces.
15. Check if the results seem to be reasonable. Create plots of pressure along the axis and in the cross-cut of the nozzle's throat (save them as a picture and data file). Create contours of pressure and Mach numbers, check if detachment occurs near to wall (using vectors or path line plots) and if critical Mach number is exceeded.
16. Save results and run calculations for reduced static pressure at the outlet (gauge pressure = 0.5369 atm). Now we expect that the Mach numbers will rise above unity downstream of the bump.
17. Compare results obtained with higher ( $p_{\text{outlet}}=0.7369\text{atm}$ ) and lower ( $p_{\text{outlet}}=0.5369\text{atm}$ ) pressures at the outlet. Verify if the *critical pressure*  $p^*$  is obtained at the throat section in the second case (use **Results/Reports** option to compute the area-weighted average of static pressure at throat section). According to theory the critical pressure is  $p^*=p_0(2/(\gamma+1))^{\gamma/(\gamma-1)}=p_0\cdot 0.528=0.9\cdot 0.528=0.4752\text{ atm}$  ( $\gamma=1.4$ ).

## *Automatic mesh adaptation*

Calculations should be made using the density-based solver with  $p_{\text{outlet}}=0.5369\text{atm}$  at the outlet.

1. In order to increase accuracy of the results in areas of big pressure gradients, turn on automatic grid adaptation (**Adapt / Mark\Adapt Cells / Gradient**). In order to do this the solution has to be initialized.

- a. Choose the appropriate grid adaptation method;

*The mesh adaptation criterion can either be the gradient or the curvature (second gradient). Because strong shocks occur inside the nozzle, the gradient is used as the adaptation criterion.*

- b. use Scale normalization,

*Mesh adaptation can be controlled by the raw (or standard) value of the gradient, the scaled value (by its average in the domain), or the normalized value (by its maximum in the domain). For dynamic mesh adaptation, it is recommended to use either the scaled or normalized value because the raw values will probably change strongly during the computation, which would necessitate a readjustment of the coarsen and refine thresholds. In this case, the scaled gradient is used.*

- c. tick Dynamic with interval equal to 100 (grid adaptation takes place after 100 iterations), based on the Static pressure;

*For steady-state flows, it is sufficient to only seldomly adapt the mesh in this case an interval of 100 iterations is chosen. For time-dependent flows, a considerably smaller interval must be used.*

2. Set Coarsen threshold to 0.3 and Refine threshold to 0.7 (tick coarsen under options first)(As the refined regions of the mesh get larger, the coarsen and refine thresholds should get smaller. A coarsen threshold of 0.3 and a refine threshold of 0.7 result in a medium to strong mesh refinement in combination with the scaled gradient.)
3. Edit controls, select coarsening and refining for the whole region, input 20000 as maximum number of cells
4. Confirm selection by clicking Apply
5. Start calculations and monitor convergence by checking mass flow plot
6. After exactly 100 iterations (as selected in point 1c) grid adaptation should occur. To check if it took place, stop calculations after about 110 iterations. Check how the grid looks. If you can see regions where mesh has been coarsened or refined it means grid adaptation is active. If not, go back to point 1.
7. Analyse results.