

Supersonic Flow Over a Wedge: Ansys Solution Outline

Pre-analysis

- See video

Geometry

We start by defining the domain of the boundary value problem. This is the fluid region around the wedge.

- Re-name Fluent analysis: "Supersonic Flow Past Wedge"
- Ensure that 2D analysis will be performed by Right Clicking Geometry > Properties > Analysis Type > 2D
- Start Discovery by Double-Clicking on Geometry
- Top Left Menu > Settings > Units and Display Precision > Length: m, Grid Spacing: .1 m
- Select Sketch > Click z-axis to select Sketch Plane > Press v to look at plane
- Select Line Tool > Click Origin (should show a green circle) > Move Along the x-axis (should be highlighted green) > Click anywhere along axis to create line of arbitrary length > Click Escape
- Select Line Tool > Click anywhere to the right of the origin on the x-axis (should be highlighted green) > Click Origin (should show a green circle) to create a line of arbitrary length
- Without leaving the Line Tool, move cursor vertically and Click anywhere above the origin on the y-axis (should be highlighted green) to create a vertical line of arbitrary length
- Without leaving the Line Tool, move cursor horizontally to the right and Click anywhere to create a perpendicular horizontal line of arbitrary length > Click Escape
- Select Dimension Tool and give the three lines the following dimensions
 - Bottom Horizontal Line: .5 m
 - Left Vertical Line: 1.3 m
 - Top Horizontal Line: 1.5 m
- Click the Line Tool > Select Right-Most Point on the bottom horizontal line > Move cursor and click to create a line at an angle
- Select Dimension Tool > Click angled line > Click x-axis > Set angle to be 15°
- Select Line Tool > Connect the open side of the geometry

- Select Constraints > Click Vertical > Click the new line to constrain it
- Enter 3D Mode to create the surface
- Click on Named Selections Tool on the bottom right > Create the following Named Selections:
 - Left and Right Vertical Edges, Top Horizontal Edge: farfield
 - Hold Ctrl to select multiple edges
 - Bottom Horizontal Edge: symmetry
 - Angled Edge: wedge

We have defined the fluid domain over which we want to solve the governing equations.

- Close Discovery
- Save project in .wbpj format

Mesh

The domain needs to be discretized in order for the solver to obtain an approximate numerical solution to the BVP and calculate the velocity components and pressure at the cell centers.

- In Workbench, Double Click on Mesh Step
- Select Mesh > Generate
- Generate a more regular mesh by selecting Face Meshing > Geometry > Select surface > Click Apply in the Details Pane
- Click Mesh > Update
- Create a Finer Mesh by selecting Sizing > Geometry > Select the Surface > Click Apply in the Details Pane > Change Element Size to 5E-2

These elements represent control volumes; the primary unknowns will be determined directly at the centers of the control volumes.

- Exit Mesher
- Save Project

Physics Setup

In this step, we define the governing equations and boundary conditions.

- Double Click Setup in Workbench
- When the Fluent Launcher opens, select Double Precision, and change Solver Processes to match the number of CPU cores on your computer > Click Start to start Fluent
- Perform Mesh Check
- Under General, change Solver Type to Density-Based

- Under Models, Double-Click Energy > Check the Energy Equation box to turn on the Energy Equation > Click Ok
- Double-Click Viscous > Select Inviscid > Click Ok

This specifies which equations need to be solved. Now, material properties and boundary conditions must be provided.

- Under Materials > Fluid > Double Click on air > Change Fluid Properties
 - Density: ideal-gas
 - Cp: 1006.43 J/(kg K)
 - Molecular Weight: 28.966 (kg/kmol)
 - Click Change/Create
- Double Click Boundary Conditions and set the following Types:
 - farfield: pressure-farfield > Gauge Pressure: 101,325 Pa > Mach Number: 3 > Click Thermal Tab > Temperature: 300 K
 - symmetry: symmetry
 - wedge: wall
- Select Operating Conditions > Change Operating Pressure to 0

Solution

Let's get the Fluent solver to determine the primary unknowns at the cell centers. The algebraic equations generated by the solver are nonlinear, so they must be solved iteratively to get the cell center values of the primary unknowns.

- Double Click Controls in the tree and ensure the Courant Number is set to 5.0
- Reduce the criteria for convergence by expanding Monitors > Double Click Residual > Reduce Absolute Criteria to 1E-6 for all fields
- Provide initial cell-center guess values by Double Clicking Initialization > Standard Initialization > Compute from: farfield > Initialize
- Start the iterations by Double Clicking Run Calculation > Number of Iterations: 100 > Click Calculate
- Monitor the Residual Plot to Ensure each data series falls below the 1E-6 tolerance

Numerical Results (Fluent)

Here we will post-process the results; all results are calculated from the cell center values of the primary unknowns.

Velocity Vectors

- From the Results Tab > Vectors > New
 - Name: velocity_vectors
 - Vectors of: Velocity
 - Color by: Velocity > Velocity Magnitude
 - Surfaces: interior-wedge_surface
 - Click Save/Display

Contour Plots

- From the Results Tab > Contours > New
 - Name: mach_contours
 - Contours of: Velocity > Mach Number
 - Surfaces: Ensure none are selected
 - Click Save/Display
- Repeat this process to create Pressure Contours
 - Name: pressure_contours
 - Contours of: Pressure > Static Pressure
 - Surfaces: Ensure none are selected
 - Click Save/Display
- Repeat this process to create Total Pressure Contours
 - Name: total_pressure_contours
 - Contours of: Pressure > Total Pressure
 - Surfaces: Ensure none are selected
 - Click Save/Display
- Repeat this process to create Total Temperature Contours
 - Name: total_temp_contours
 - Contours of: Temperature > Total Temperature
 - Surfaces: Ensure none are selected
 - Click Save/Display

Mach Number Plot

- From the Results Tab under Surface > Create > Line/Rake
 - Name: plot_line
 - Type: Line > End Points: (0, .4), (1.5, .4)
 - Click Create

- From the Results Tab under Plots > XY Plot > New
 - Name: mach_plot
 - Y Axis Function: Velocity > Mach Number
 - X Axis Function: Direction Vector
 - Surface: plot_line
 - Click Save/Plot
- To get more sample points, Under Results in the Tree > Expand Surfaces > Right Click plot_line > Edit
 - Change Type: Rake
 - Number of Points: 100
- Redraw the Plot by Expanding Plots in the Tree > Right Click on mach_plot > Edit > Save Plot

Make sure to save the project

- If prompted, select Save for Current and Future Calculations to preserve your work

Verification and Validation

To verify the solution, refine the mesh to ensure the results are independent of the discretization.

- In Workbench, Right Click on the Fluid Flow (Fluent) Part of the Module > Duplicate
- Name the new Module: "Mesh Refinement"
- Double-Click on Solution to Open Fluent
- Generate an initial solution as before by Double Clicking Initialization > Compute From: farfield > Initialize
- Double-Click Run Calculation > Ensure Number of Iterations is 100 > Click Calculate to get a solution

We want to refine the mesh in the area of the shock.

- Under the Domain Tab, in the Adapt Section > Click Manual
- Select Cell Registers > New > Field Variable
 - Name: pressure_grad
 - Type: Cells Inside Range
 - Derivative Option: Gradient
 - Gradient of: Pressure > Static Pressure
 - Gradient Minimum: 10,000
 - Gradient Maximum: 110000
 - Click Save-Display

- Ensure that this range covers the shock by expanding Contours in the tree, Right Click on mach_contours > Add to > Active Window
- In the Manual Mesh Adaptation window, Click the arrow next to Refinement Criterion > Select pressure_grad > Click Adapt
- View the new Mesh by Double Clicking General in the tree > Under Mesh, select Display > Click Display in new window

The calculation process can then be run again.

- Double Click Run Calculation > Ensure Number of Iterations is set to 100 > Calculate

The residuals should end up converging to the desired value. If convergence issues occur and the residuals appear to oscillate continuously instead of decreasing, try the following:

- Double Click Methods > Flow > Third-Order MUSCL
- Double Click Run Calculation > Change Number of Iterations to 200 > Calculate

Right Click on mach_contours in the tree > Display. The new solution should show the shock more distinctly.

Make sure to save the project.