

Flat Plate Flow: Fluent Solution Outline

Rajesh Bhaskaran

Cornell University

Problem specification

- See tutorial for problem specification <https://confluence.cornell.edu/x/9YxoBQ>

Pre-Analysis

- See tutorial for
 - Mathematical model
 - Numerical solution strategy
 - Hand calculations of expected results

Geometry

- Start Workbench
- Select Fluid Flow (Fluent) analysis system by dragging
 - Rename: Flat Plate, ReL=10,000
- Geometry > Properties > 2D
- Geometry > New SpaceClaim Geometry
- Cancel any firewall warnings
- File > SpaceClaim options > Units > m
 - Minor grid spacing: 0.1 m
- Select xy plane
- Sketch rectangle
- Dimension: Length = 1 m, Height = 0.5 m
- Switch to 3D mode
- Rename Design1 in tree to Flat_Plate_Domain
- Domain for the BVP is defined at this point
- Exit SpaceClaim
- Save project in wbpj or wbpz format

Mesh

- In this section, we divide the domain into cells or control volumes

- Project window > Mesh
- Try default mesh
- Need more regular mesh: use Face Meshing
- Edge sizings: 50x100
 - Use hard setting
 - Use bias factor of [] for vertical edges
 - Check height of cells near the wall
- Check mesh statistics
 - 5000 elements (cells)
 - We have marked out the locations at which the solver needs to calculate the primary unknowns
 - How many algebraic equations will the solver have to generate?
- Specify named selections: farfield1, farfield2, farfield3, plate, bvp_domain
- Check named selections in graphics window
- Exit mesher
- Update Mesh cell (in project page)
- Save project

Model Setup

- In this section, we specify the BVP
- Start Fluent from Setup
- Select Double precision
- Display mesh
- Check mesh
- Specify material properties: $\rho = 1 \text{ kg/m}^3$, $\mu = 1\text{e-}4 \text{ kg/(m s)}$
- Governing equations are defined
- Boundary conditions
 - farfield1 and farfield2: velocity-inlet, 1 m/s in x-direction
 - farfield3: pressure-outlet, $p=0 \text{ Pa}$
 - This is gauge pressure
 - Check operating pressure
 - plate: wall
- BVP is completely defined at this point

Numerical Solution

- In this section, we get the Fluent solver to solve the BVP
- Specify initial guess at all centers

- $u = 1\text{m/s}$
- $v = 0\text{ m/s}$
- $p = 0\text{ Ps}$
- Decrease residuals criteria (i.e. tolerance) to $1\text{e-}6$
- Set number of iterations = 100
- Iterate to convergence

Numerical Results

- Start CFD-Post from Results
- Velocity vectors
 - Symbol size = 0.2
 - Check boundary layer development
 - Is the velocity range plausible?
- Velocity magnitude contours
 - Increase # of contours to 101
 - Notice boundary layer development and thickness
 - Check values using probe and figure out the approximate boundary layer height at $x=L$
 - Are the boundary conditions on velocity satisfied?
 - Save the plot using Camera icon or snipping tool
- Pressure contours
 - Increase # of contours to 101
 - What's the effect of the boundary layer on pressure?
 - Is the gauge pressure range plausible?
 - Is the boundary condition on pressure satisfied?
 - Contrast pressure and velocity variations
- Velocity profiles
 - Create lines at $x = 0.4\text{ m}$ and $x = 0.8\text{ m}$: Location > Line
 - Create velocity profile plot using Chart option
 - Adjust y-axis range to be 0-0.2m
 - Compare boundary layer thickness to expected value from boundary layer theory
 - Create non-dimensional variables using Expressions: $\frac{y}{L}$ and $\frac{u}{U_\infty}$
 - Replot in terms of non-dimensional variables
 - Prettify plot
- Rescale velocity profiles to check similarity
 - Duplicate Velocity profiles object in tree
 - Rename: Velocity profiles scaled

- Create expression: $yscaled_exp = Y/\sqrt{X}$
- Create $yscaled$ variable from this expression
- Change vertical axis from Y to $yscaled$
- Do the velocity profiles collapse to a single curve in the boundary layer?
- Extend to compare the scaled velocity profiles to the Blasius solution
- Export data to csv file if necessary (to replot in Excel)
- Plot profiles of $\frac{\partial u}{\partial x}$ and $\frac{\partial u}{\partial y}$
 - Fluent > Run Calculation > Data File Quantities > dX-Velocity/dx (and other velocity derivatives)
 - Run 1 iteration to transfer to CFD-Post
 - Plot $\frac{\partial u}{\partial x}$ and $\frac{\partial u}{\partial y}$ at $\frac{x}{L} = 0.8$
 - Do their relative magnitudes agree with assumptions in boundary layer theory?
- Calculate drag coefficient
 - Go back to Fluent window
 - Set Reference Values
 - Calculate drag coefficient under Reports

Verification and Validation

- Mesh refinement
 - Double number of divisions in both directions
- Check effect of moving the outer boundaries

Flat Plate Convection: Fluent Solution Outline

Problem specification

- See tutorial for problem specification <https://confluence.cornell.edu/x/ZDLEFg>

Pre-Analysis

- See tutorial for
 - Mathematical model
 - Numerical solution strategy
 - Hand calculations of expected results

Geometry

- Same as case without heating

Mesh

- Same as case without heating

Model Setup

- In this section, we specify the modified BVP
- Start the ANSYS project from the case without heating
- In Workbench, duplicate project from Results
 - Rename: Convection, ReL=10,000 Pr=1
 - Note Geometry and Mesh are shared
- Start Fluent from Solution
- Check current solution by looking at velocity contours
- Rerun to convergence
 - Drop all Residual Criteria to 1e-6
 - We have recreated previous solution without heating
- Turn on energy equation under Models
- Input additional material properties
 - $C_p = 1e4 \frac{J}{kg K}$
 - $k = 1 \frac{W}{m K}$

- Input additional boundary conditions
 - farfield1 and 2: Temperature = 400 K
 - plate: Temperature = 300 K
- Modified BVP is completely defined at this point

Numerical Solution

- In this section, we get the Fluent solver to solve the BVP
- Reinitialize
- Iterate to convergence

Numerical Results

- Start CFD-Post from Results
- Velocity magnitude contours
 - Should be in the tree from case without heating
 - Note velocity field hasn't changed. Why?
- Temperature contours
 - Increase # of contours to 101
 - Try changing units to K under Edit > Options
 - Doesn't seem to work
 - Workaround: Create user-defined variable
 - Notice thermal boundary layer development and thickness
 - Compare to velocity boundary layer by plotting temperature and velocity magnitude contours side-by-side
 - Go back to single pane
 - Are the boundary conditions on temperature satisfied?
 - Use probe to check
- Temperature profiles
 - Copy *Velocity profiles* object and modify
 - Create non-dimensional user-defined variable: $T_{nondim} = \frac{T_w - T}{T_w - T_\infty}$
 - Replot in terms of T nondim
 - Prettify
- Nusselt number plot
 - Transfer h from Fluent to CFD-Post
 - Fluent > Reference values > Temperature = 400 K
 - Fluent > Data File quantities > Surface Heat Transfer Coefficient
 - Make sure previous selections were not unselected

- Run 1 iteration
- Refresh CFD-Post when prompted
- Create line corresponding to plate: Location > Line
- Plot h vs x
- Create user-defined variable for Nu_x
- Create user-defined variable for Re_x
- Change plot to Nu_x vs Re_x
- Export to Excel
- Curve fit in Excel