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 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: ρ = 1 kg/m3, μ = 1e-4 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 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 = 1m/s
- v = 0 m/s
- p = 0 Ps

- Decrease residuals criteria (i.e. tolerance) to 1e-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 $x = 0.4\ m$ and $x = 0.8\ m$: 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