Direct Numerical Simulation of Cavity Flow

Ohio Space Grant Consortium
Joe Mileski
Ohio Northern University
Direct Numerical Simulation

• Does not use a turbulent Model
  • Treats all flow as laminar
  • Grid spacing is tight enough to capture eddies as if laminar, using length scale
Cavity Flow

• Introducing a cavity, hole, in a standard geometry

• Used a rectangular cavity, in two dimensions to lessen computational timing.
Determining the Geometry

• Ratio of depth to length of the cavity
  • 1:4 was used to compare to Pope and Hardin [1]

• This characteristic length was used to calculate the Reynolds number [2]
Determining the Length Scale

- Need to capture the smallest eddy in turbulence possible [1]
  - Length scale is a function of dissipation rate and kinematic viscosity.
    - Dissipation rate is a function of viscosity, density, Reynold’s number and characteristic length

- Using a Reynolds number of 5000 as used by Pope[2]
  - Length scale was determined to be 0.0135
  - Dividing by five, lowers grid spacing to 0.0027
2-D vs. 3-D

• Saves Computational Time
  - Two Dimensional have roughly 10% of the 3-D cells
• 2-D has all the characteristic lengths needed to determine Reynolds number and length scales.

• Has limitations, such as difference in flow, as movement in third direction is neglected [3]
Preprocessing

- Due to use of direct numerical simulation, structured grid was selected.
- Cavity was given a spacing of one length scale previously mentioned.
- Cavity floors started with length scale spacing, and expanded using an hyperbolic-tangent growth rate.
- Vertical flow walls were set to capture flow, while reducing computational timing.
- Upper horizontal flow wall was determined based on the number of nodes from three lower edges.
Boundary Conditions

• The cavity itself
  • Adiabatic No-Slip Walls

• The Floor
  • Slip Walls

• The Rest of the Flow-Box
  • Modified Riemann Sum Farfield
• Mach number was determined by Reynold’s number, though was slightly higher due to concern flow may have been too low.

• Flow equations were set to Navier-Stokes Laminar flow
  • Turbulence model was neglected, as it was not used.

• Output results every twenty iterations after 2000.
  • Repeated until convergence was acceptable
The Courant-Friedrichs-Lewy (CFL) Number was then lowered to increase stability increments of 10 for an additional 1000 iterations.
Velocity in in/s
Comparing to previous work

Pressure solved by me

Pressure solved by Pope [1]
For Continued Research

• Add a third dimension
  • Will need time on a super-computer

• Run several different geometries, using 3-D DNS and Turbulent NS equations to validate turbulent models.
References


