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

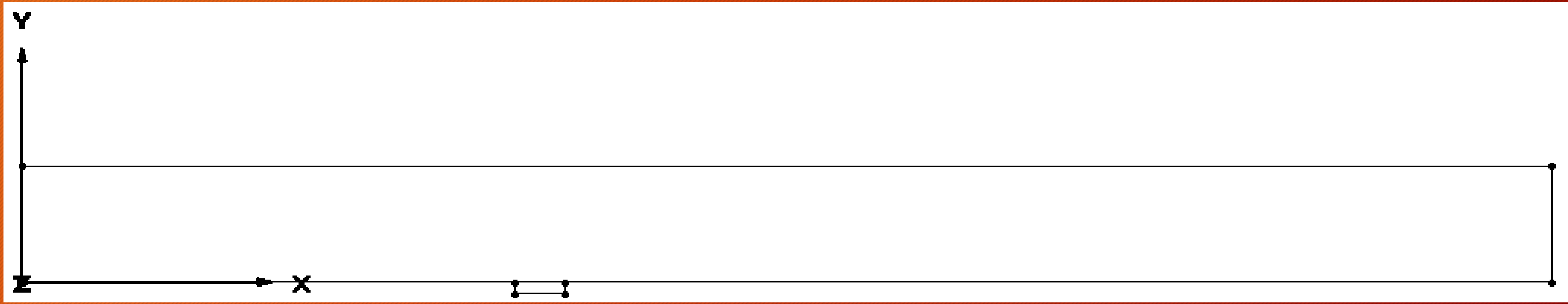
Cavity Flow

- 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

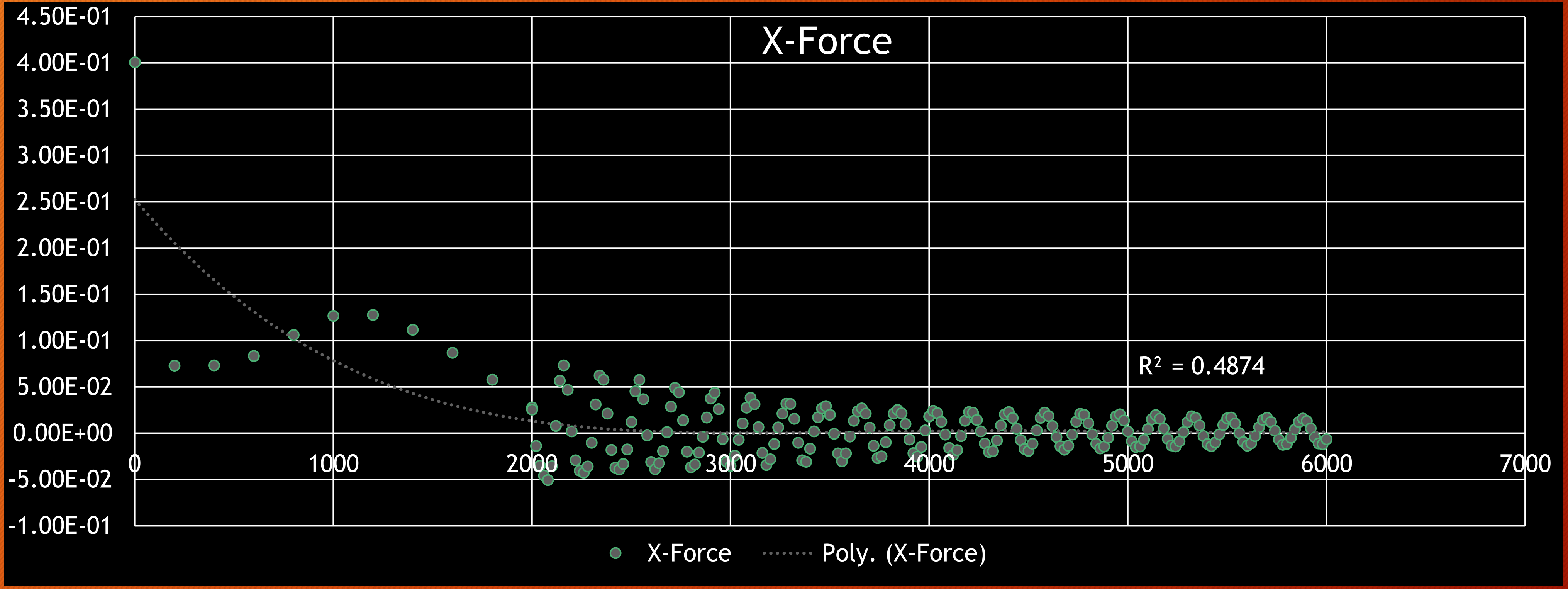
Boundary Conditions

- The cavity itself
  - Adiabatic No-Slip Walls
- The Floor
  - Slip Walls
- The Rest of the Flow-Box
  - Modified Riemann Sum Farfield

Job File

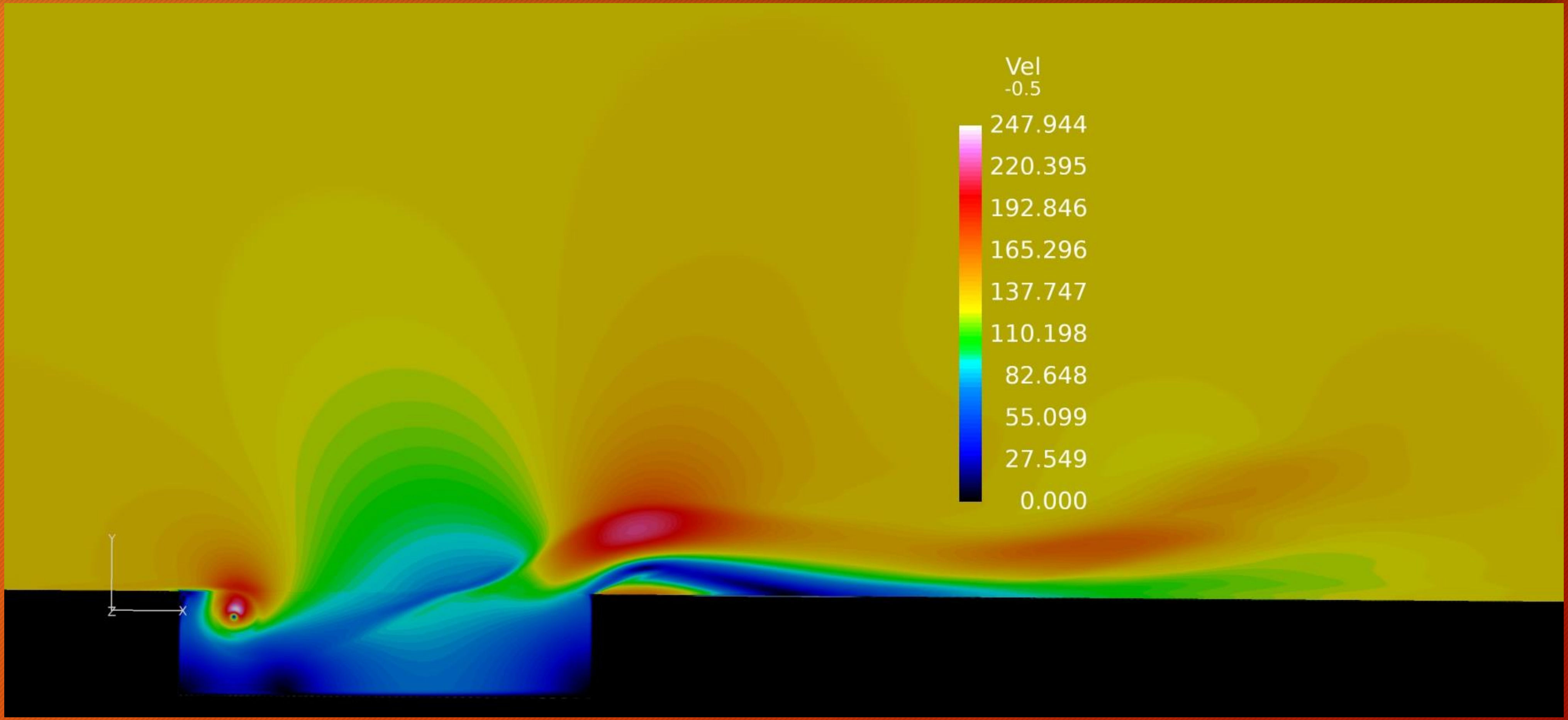
- Mach number was determined by Reynold's number, though was slightly higher due to concern flow may have been too low.
- Flow equations were set to Navier-Stokes Laminar flow
  - Turbulence model was neglected, as it was not used.
- Output results every twenty iterations after 2000.
  - Repeated until convergence was acceptable

X-Force Convergence

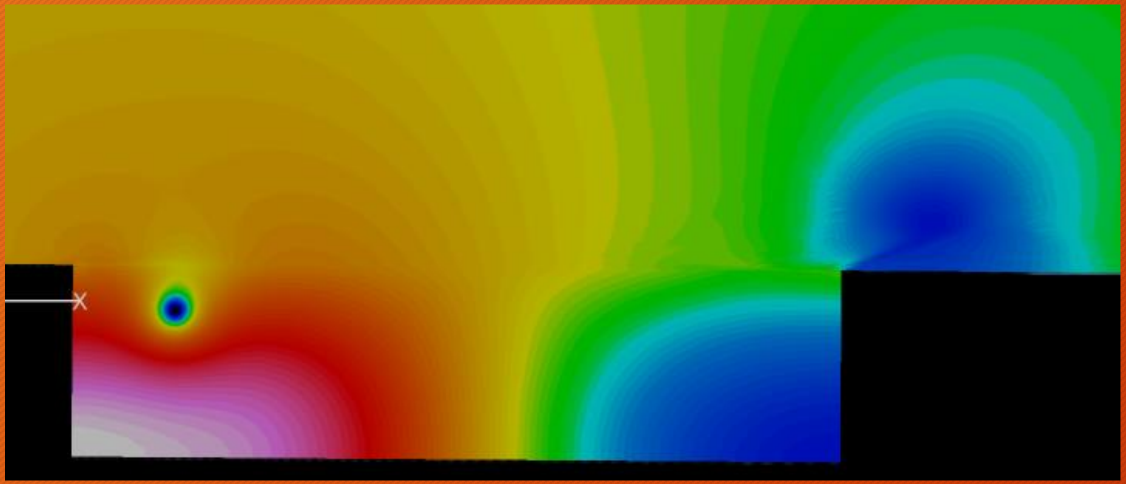


- The Courant-Friedrichs-Lewy (CFL) Number was then lowered to increase stability increments of 10 for an additional 1000 iterations

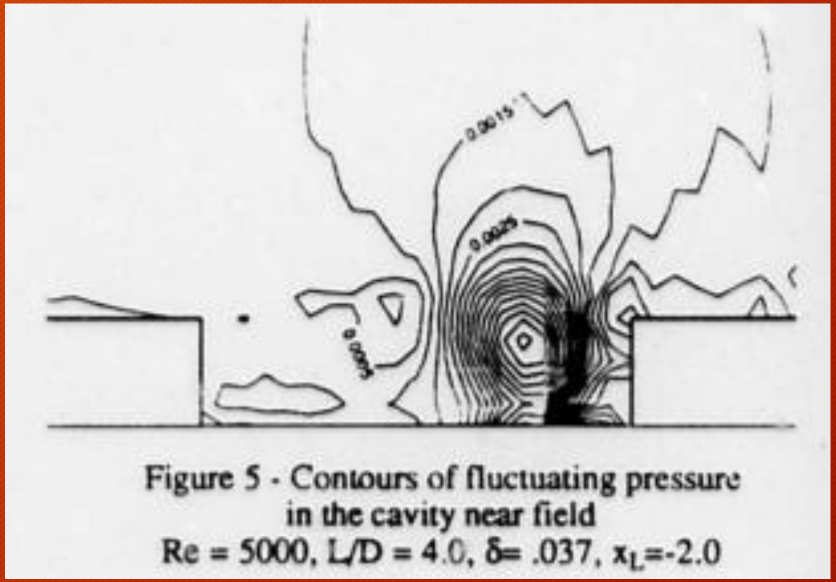
Velocity in in/s



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

- [1] D. S. Pope and J. C. Hardin, "A Numerical Investigation of Two-Dimensional Cavity Flow," *SAE Technical Paper Series*, Jan. 1995.
- [2] H. Tennekes and J. L. Lumley, *A first course in turbulence*. Cambridge, Mass. u.a.: MIT Press, 1985.
- [3] W. F. J. Olsman and T. Colonius, "Numerical Simulation of Flow over an Airfoil with a Cavity," *AIAA Journal*, vol. 49, no. 1, pp. 143–149, 2011.
- [4] *Cobalt Manual*, *Version 8*, Cobalt Solutions, LLC, Dayton, OH, 2018.