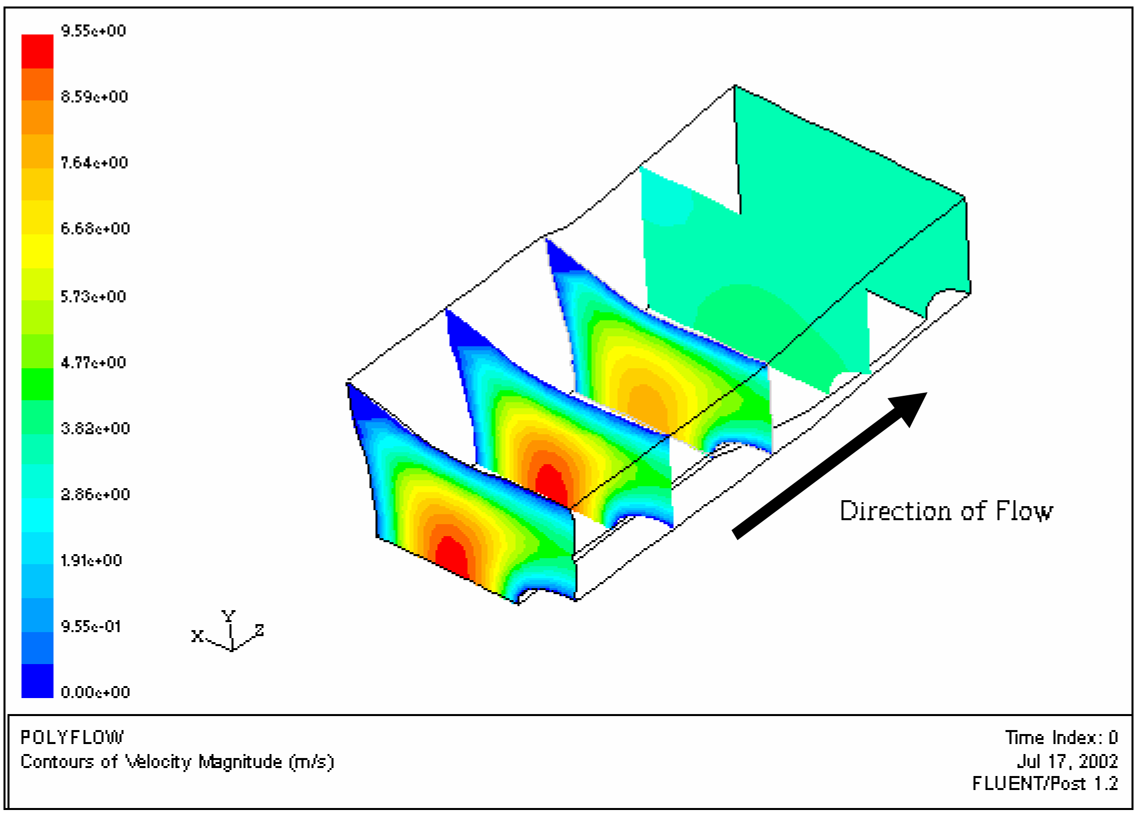
NORTHERN ILLINOIS UNIVERSITY
Department of Mechanical Engineering
DeKalb, IL 60115

A Finite Element Simulation of Flow of Polymer in an Extrusion Die Using PolyFlow®

A Preliminary Report
July 18, 2002



Prepared by
Srinivasa Rao Vaddiraju, M.S. Candidate
Mechanical Engineering, NIU
Under Guidance and Supervision of Prof. M. Kostic
www.kostic.niu.edu/extrusion

Summary

In this report I present the results from Finite Element Analysis of flow of a simple Power Law model fluid through a profile extrusion die. The analysis was performed using Fluent's Polyflow, a CFD software. The desired extruded profile is a rectangular section with a single circular hole and "inverse extrusion simulation" was performed to obtain the desired die cross section as output.

The geometrical model was built and meshed appropriately using Fluent's Gambit, a preprocessing software for modeling and meshing. The whole Program is run using Polyman interface and starting Gambit, Polydata, Polyflow and Flpost from it. The boundary conditions and material properties were given as input using Fluent's Polydata. The flow analysis was performed using Fluent's Polyflow solver, a software for modeling extrusion processes and extrusion dies. The results were visualized using Fluent's Flpost, a postprocessor interface.

Problem Description

A polymer under pressure is transferred through die cross section from inlet to outlet.
The problem is a viscous, incompressible steady-state flow simulation. The desired output profile of the extrudate in this case is a rectangular cross section with a circular hole at the center. The inlet profile is irregular cross section with an elliptical-like hole at its center so as to give the desired output shape, see also <www.kostic.niu.edu/extrusion>.
The die geometry is symmetric about a vertical plane and a horizontal plane passing through the middle of the die. Therefore, I modeled only a quarter of the domain of the die.

Material Data

The material properties of the polymer were taken as a simple Power-law model non-Newtonian, viscous, incompressible fluid. The viscosity of this polymer is given as a function of shear rate represented using the Power-Law model.

$\eta = K'\cdot(\lambda\cdot\gamma)^n = (K'\lambda^n)\gamma^n = K\gamma^n$ where $K=K'\lambda^n$

> where, K is the consistency factor, n is the power-law index, which is a property of a given material, and λ is the fluid time characteristic.

K' is taken as 30 units

λ is taken as 1 and

n is taken as 0.75

Process Conditions

To set up the finite element model five types of boundary condition data were taken: inlet, solid wall, free surface, symmetry surface and outlet. For each of these boundaries the following data was specified:

- Inlet: Flow rate = 50 units
- Die walls: All the components of the velocity were set to zero(no slip condition).
- Free deforming surfaces of the extruded profile: Pressure = 0 and Normal velocity =0
- Symmetry surface: Normal velocity = 0 and Tangential force = 0
- Outlet: Normal force= 0 (may be arbitrary), and Tangential force = 0 (uniform velocity profile).

Results of Extrusion Simulations

After the simulations were done in Polyflow using the material properties and boundary conditions specified in Polydata, the results were viewed using Flpost postprocessor.

To present results like polymer pressure field and velocity field within the die, several iso-contour plots were extracted from the obtained solution. In the figures, we can see the pressure distributions and velocity distributions on the outer surface of the domain, and along symmetry plane and several cross-sections perpendicular to flow direction.

Concluding Remark:

This is just a simple example as illustration of our initial work to learn and evaluate PolyFlow Software. Activites are underway to simulate more complex extrusion processes and materials, with an objective to simulate Scintillator extrusion processes and die designs for Fermi Accelerator Lab.

Note: Some of the previous simulations done in ANSYS-Flotran are attached as appendix A.

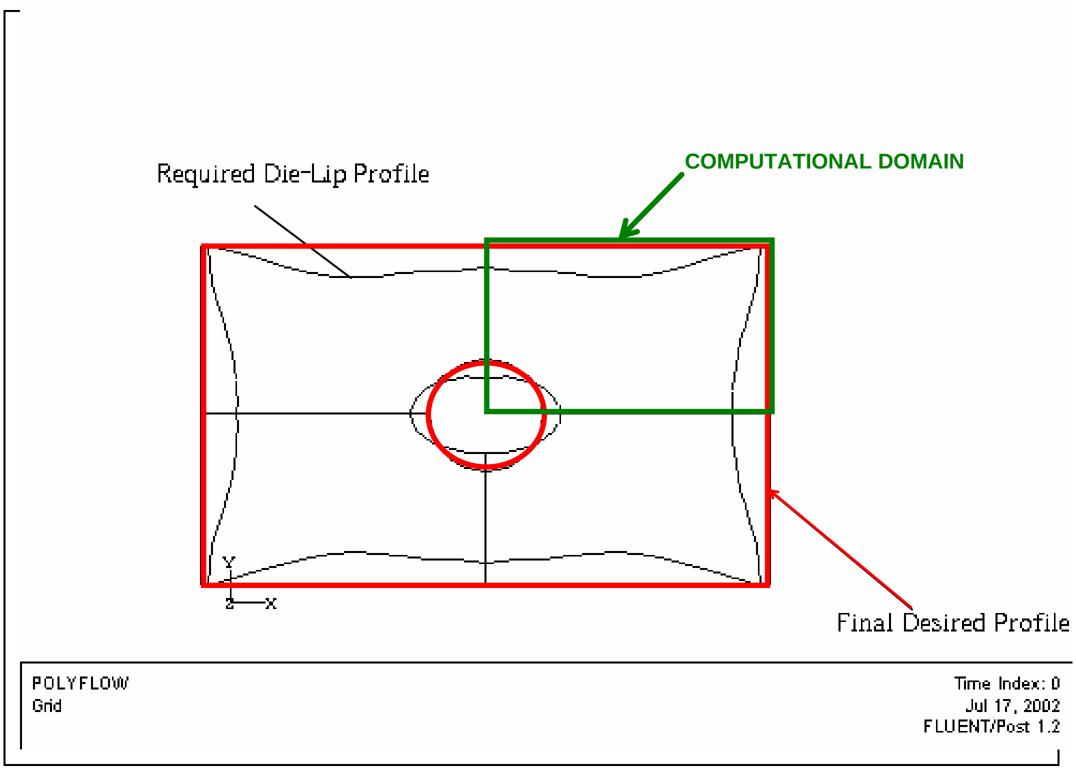


Fig.1: Display of full domain of a rectangular profile with a center hole

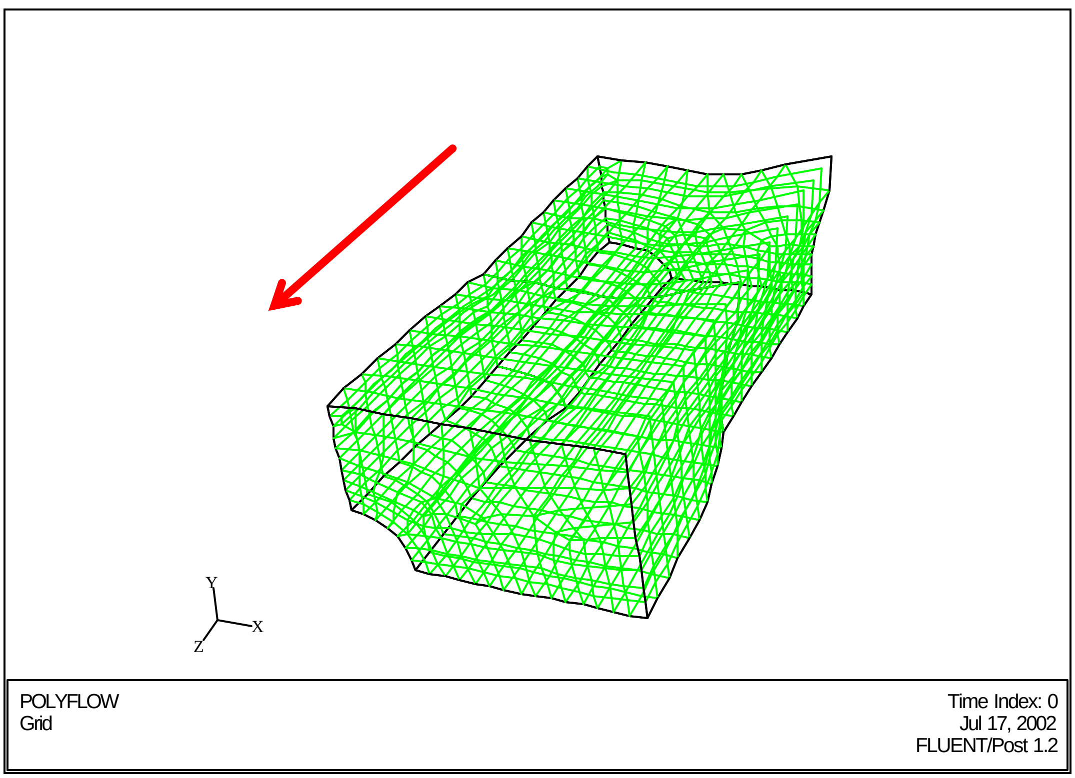


Fig.2: Finite element mesh (for die-profile and free-surface flows) generated in Gambit

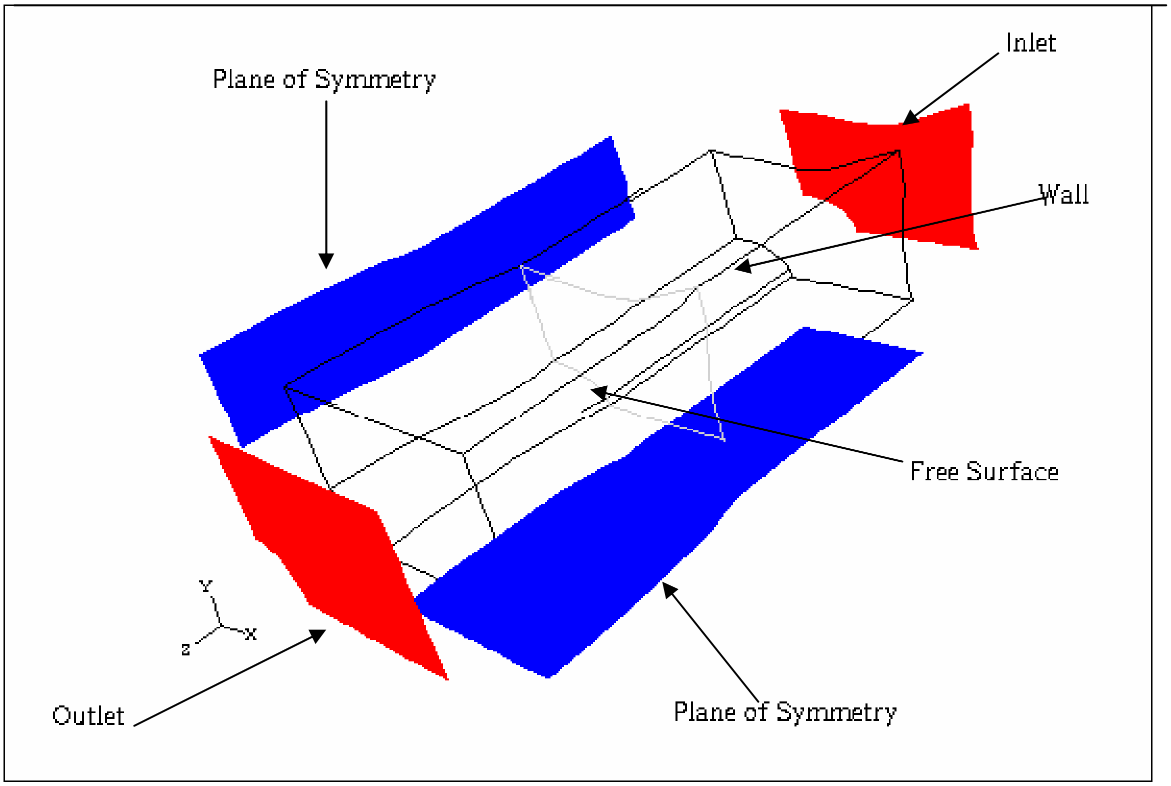


Fig.3: Boundary surfaces on a quarter (computational) domain

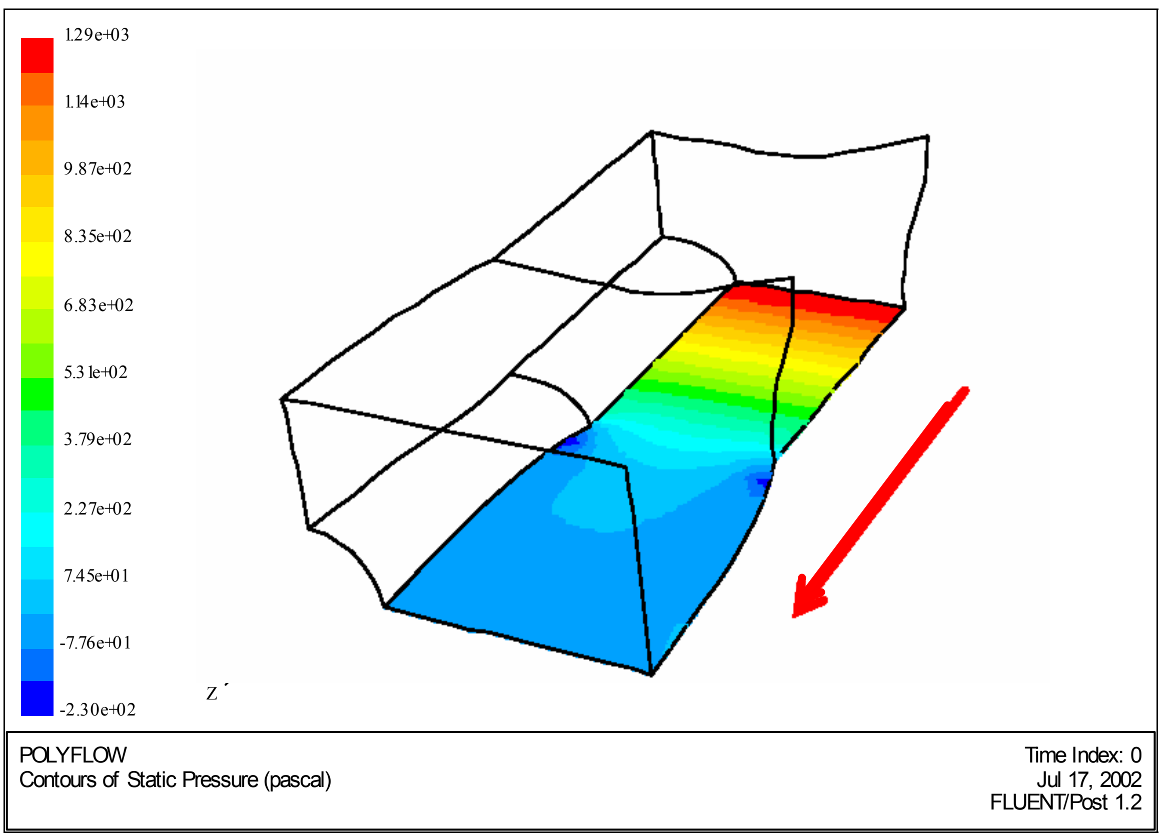


Fig.4: Pressure distribution along one center-plane in flow direction

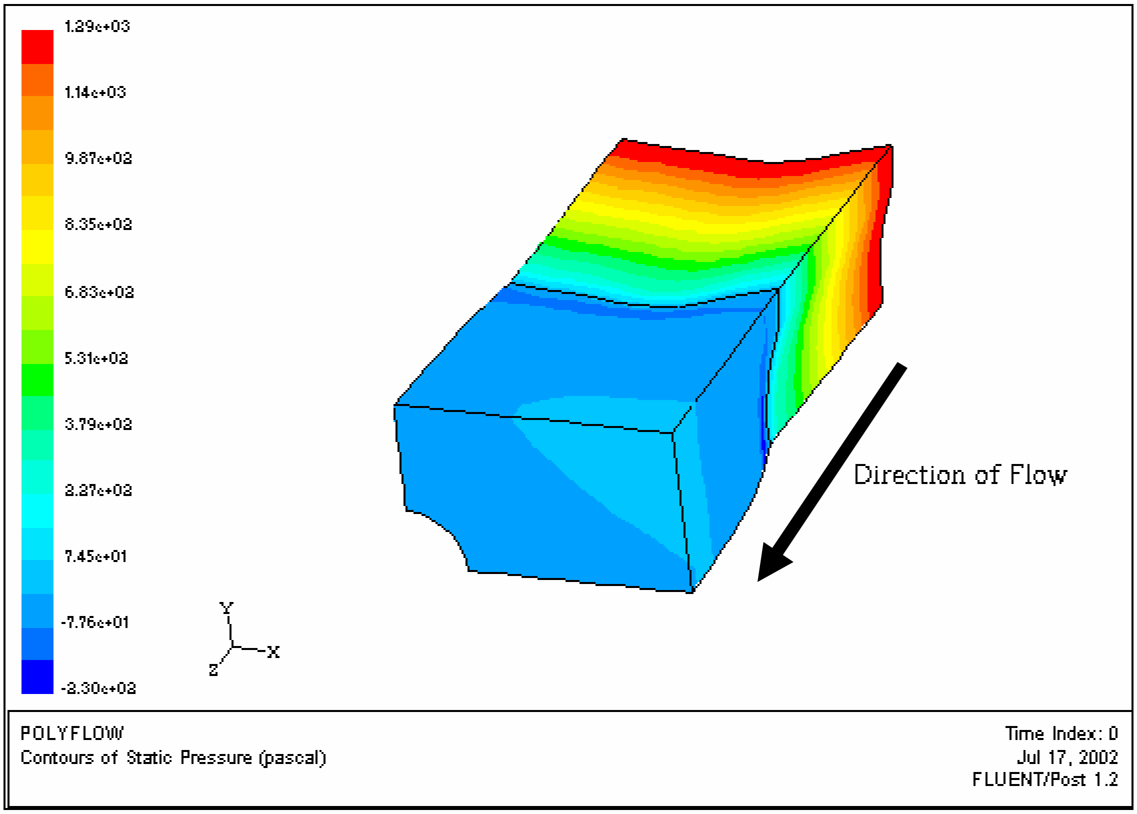


Fig. 5: Pressure distribution on domain outer surface

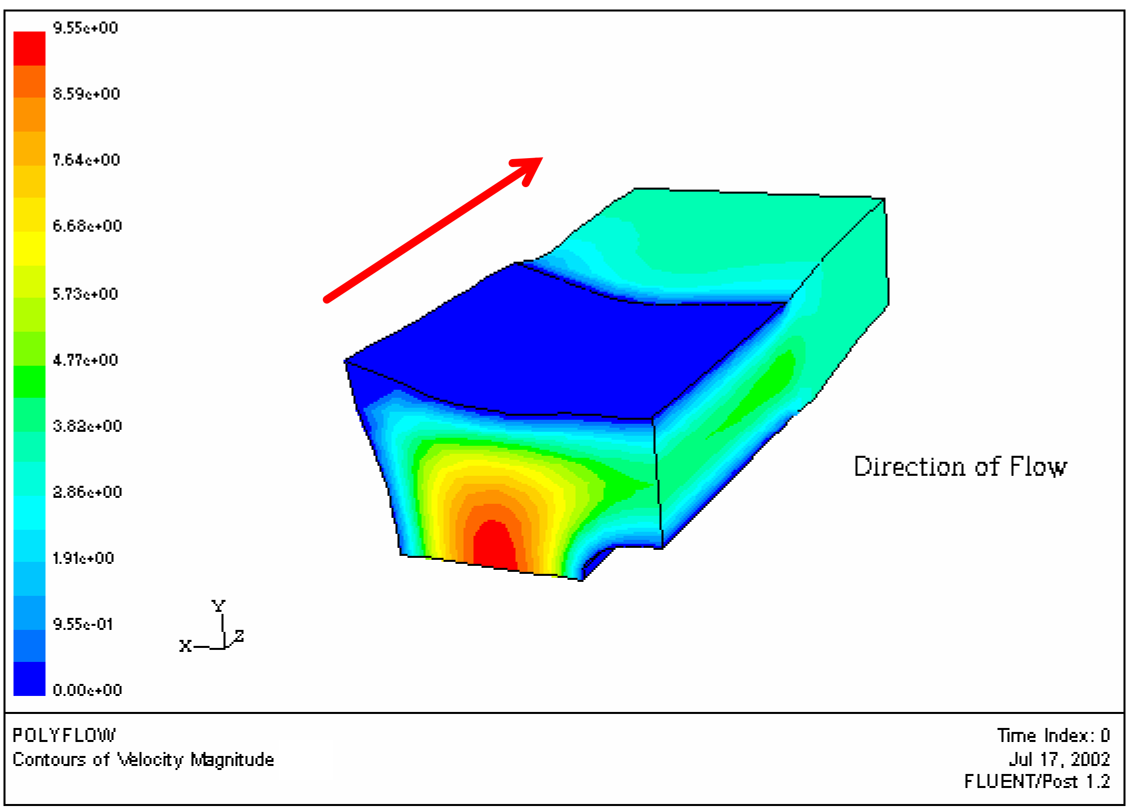


Fig.6: Velocity distribution on domain outer surface

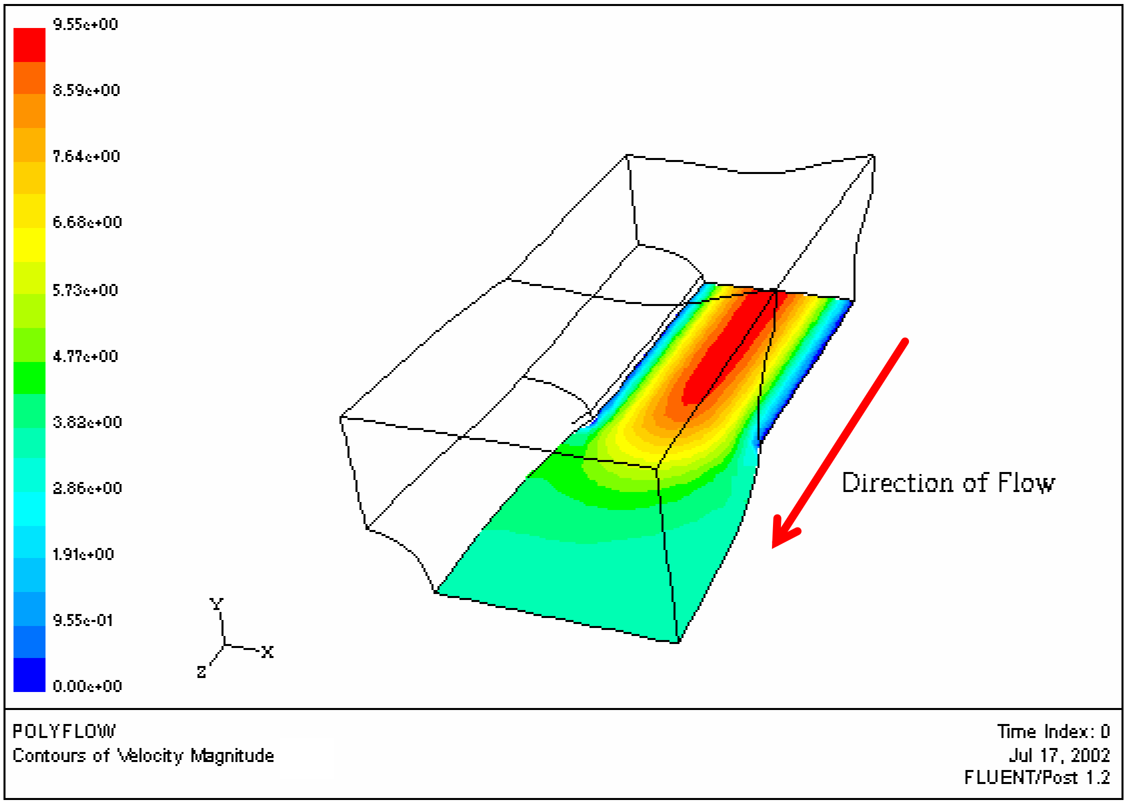


Fig.7: Velocity distribution along one center-plane in flow direction

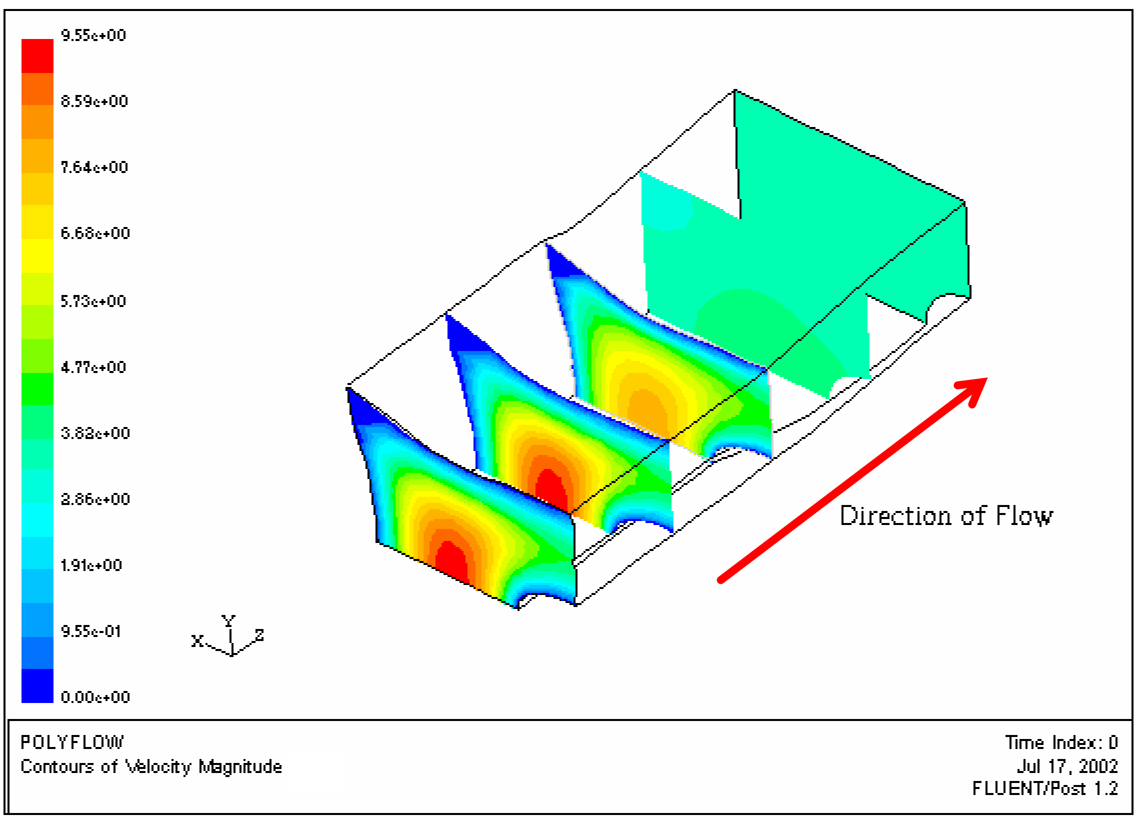


Fig.8: Velocity distribution on different cross-sections perpendicular to flow direction

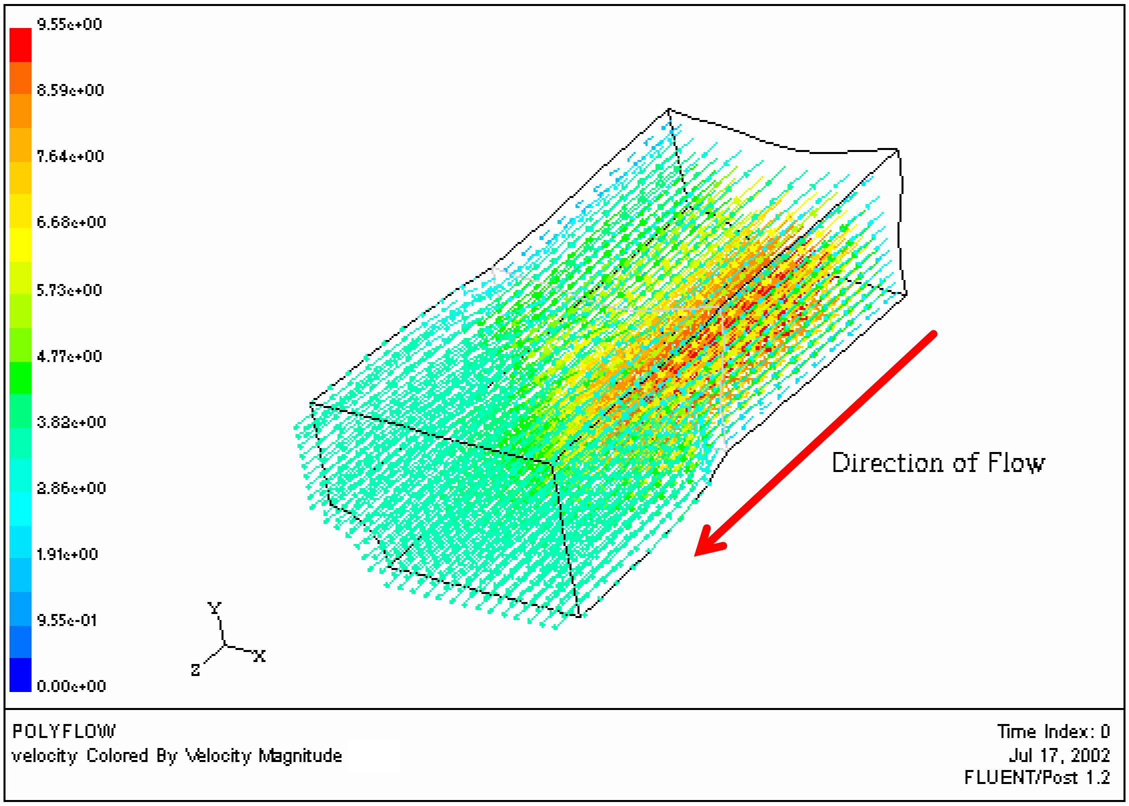


Fig.9: Domain velocity vectors colored by velocity magnitude

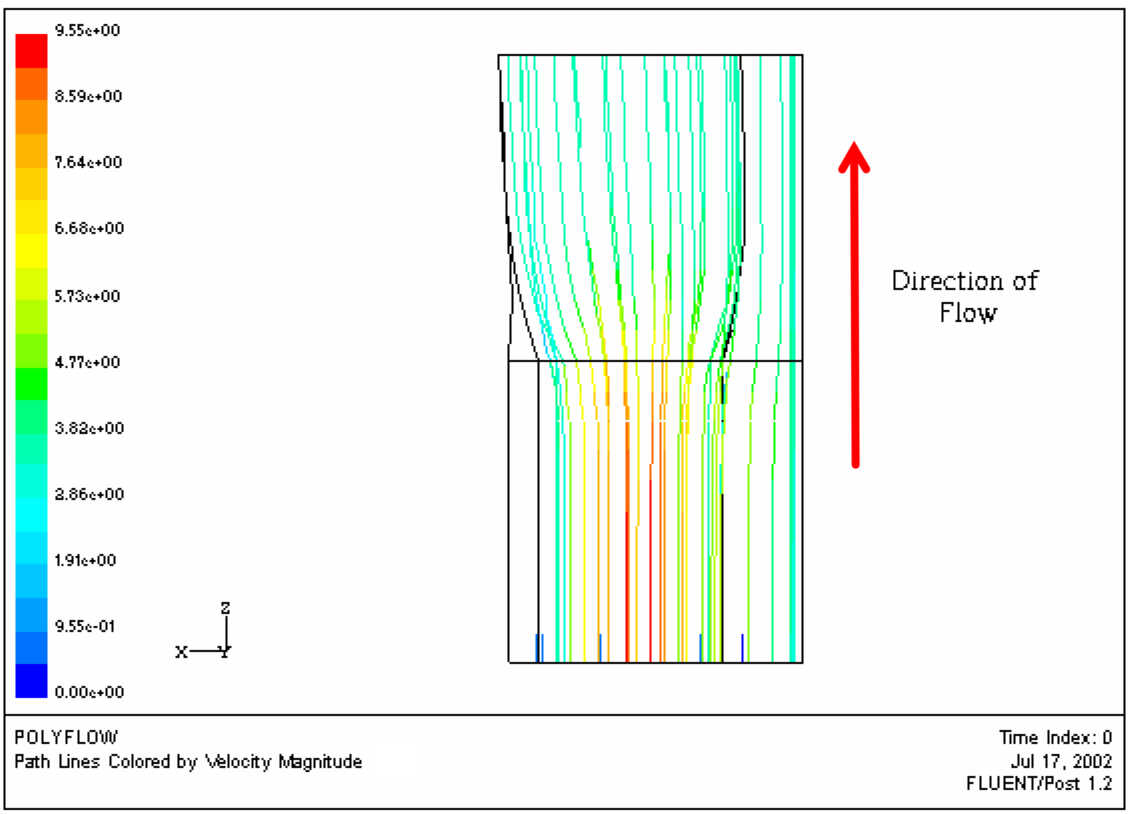


Fig.10: Particle traces from top colored by velocity magnitude

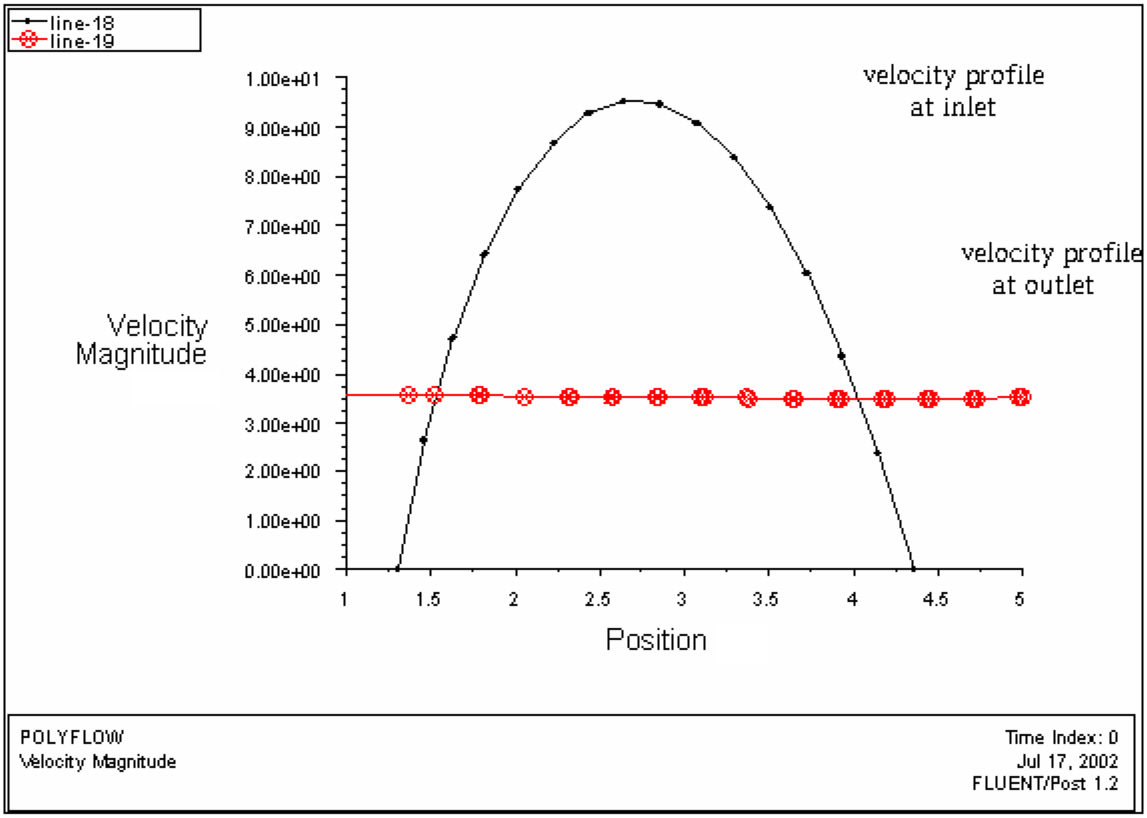


Fig.11: Velocity profile at inlet and outlet for one quarter of domain

APPENDIX A

Another CFD simulation using ANSYS Flotran software

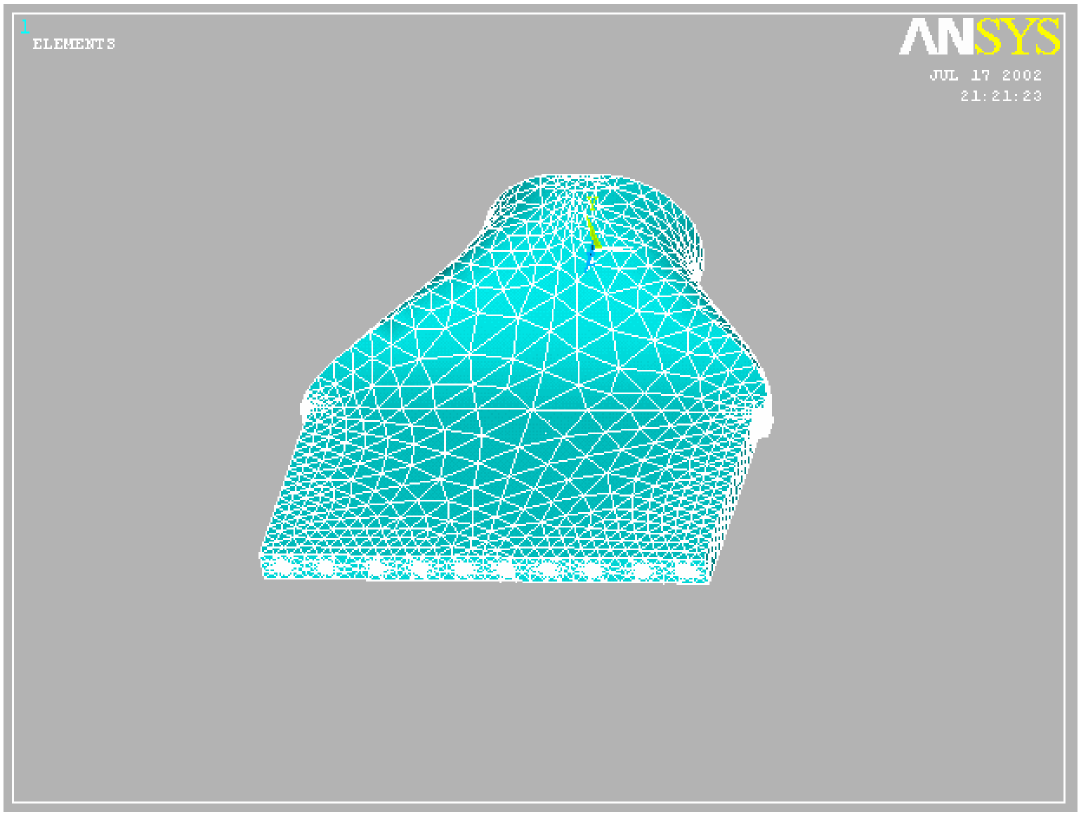


Fig.A1: Finite element mesh

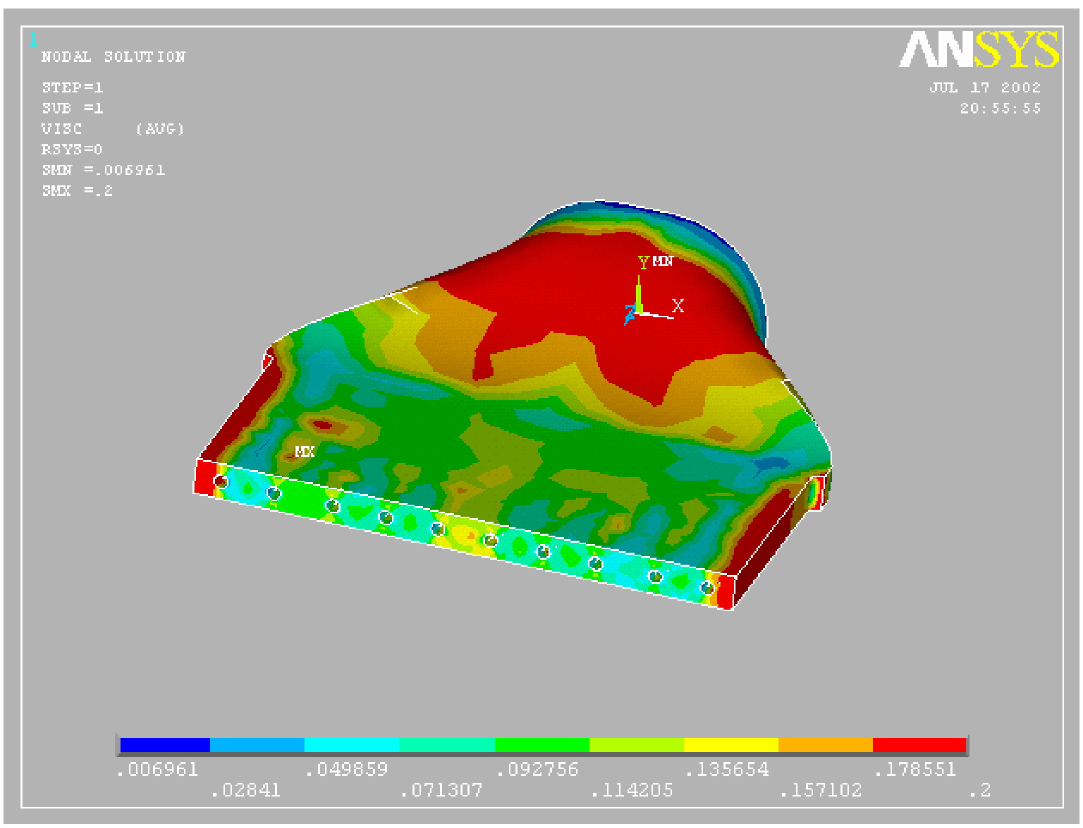


Fig.A2: Polymer viscosity on domain outer surface

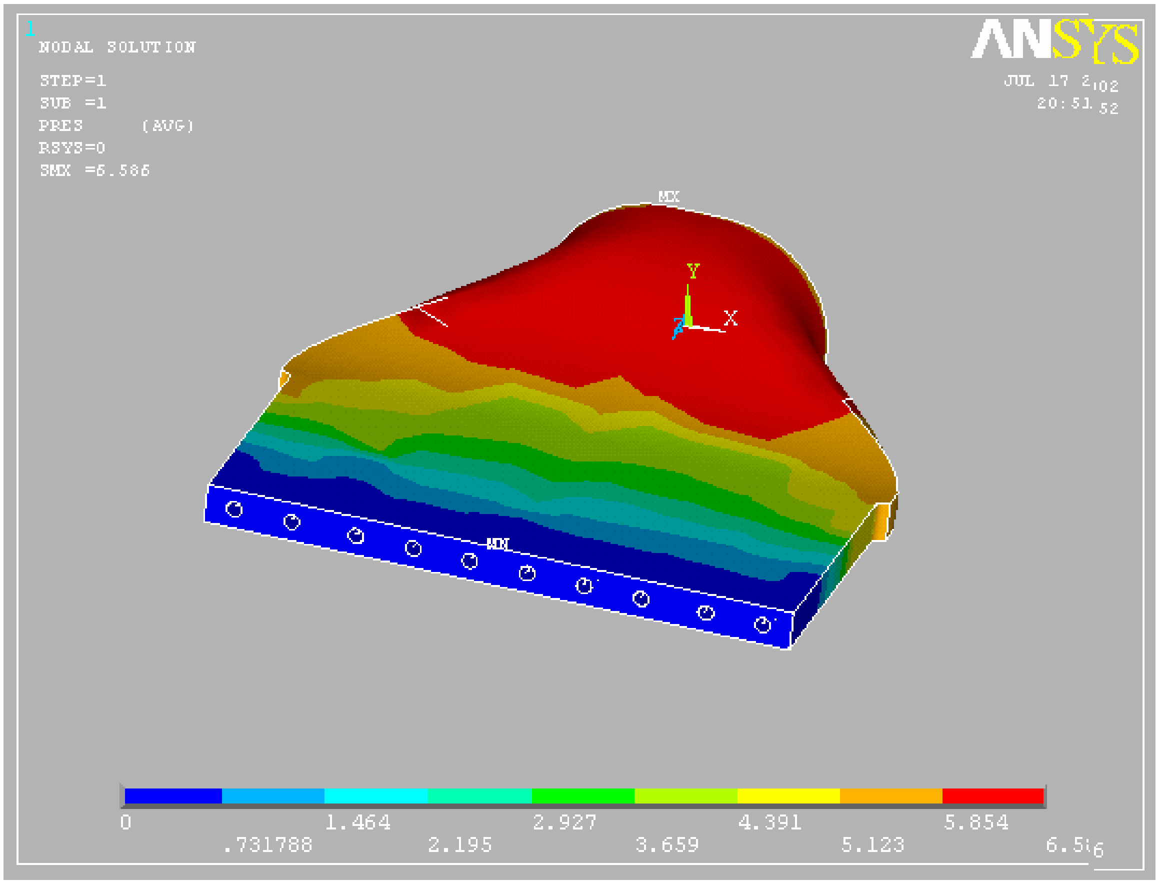


Fig .A3: Pressure distribution on domain outer surface

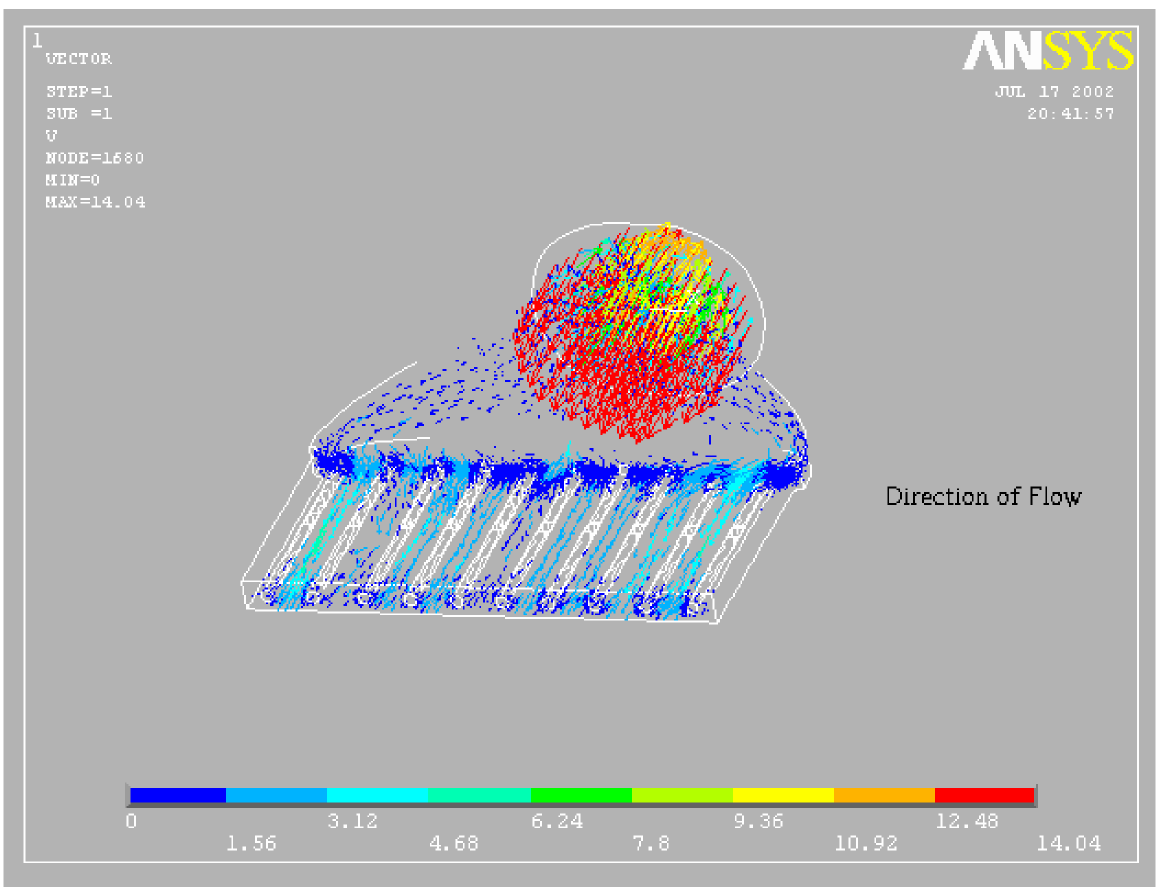


Fig.A4: Velocity vectors (colored by magnitude)