

## **COMPUTATIONAL FLUID DYNAMICS SIMULATION FOR IMPROVED TURBULENT FLOW PREDICTION**

*Jaisri. A<sup>1</sup>, S. Prabhakaran<sup>2</sup>, & D. Sundar<sup>3</sup>*

*<sup>1</sup>M E-Engineering Design, Paavai Engineering College, Namakkal*

*<sup>2</sup>Professor of Mechanical Department, Paavai Engineering College*

### **ABSTRACT**

*In this paper, the flow of air through a Pressure Regulating and Shut-Off Valve (PRSOV) has been solved numerically with the objective of determining the pressure of the PRSOV pilot chamber. The flow through the valve is transient, compressible, and turbulent in nature. The flow has been solved using ANSYS FLUENT coupled with a special User Defined Function (UDF). The UDF is used for dynamic meshing, automatic data acquisition from the ANSYS FLUENT solver, and scheduling the numerical test procedure. Within the UDF, the valve inlet pressure is varied in a stepwise manner. For every value of inlet pressure, transient analysis leads to quasi-static flow through the valve. Spool forces are calculated based on different pressures at the inlet. From this information of spool forces, the pressure characteristic of the pilot chamber of the passive control circuit has been derived. The same characteristics have also been obtained after modeling the flow analytically. Both results have been compared. It is observed that CFD analysis of the flow has led to improved results.*

**KEYWORDS:** *Fluid Dynamics*

---

### **Article History**

**Received: 03 May 2024 | Revised: 08 May 2024 | Accepted: 10 May 2024**

---

