Computers & Fluids 101 (2014) 233–240

Contents lists available at ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

Dynamic simulation of a pressure regulating and shut-off valve

Binod Kumar Saha [a], Himadri Chattopadhyay [b,*], Pradipta Basu Mandal [a], Tapas Gangopadhyay [a]

[a] Product Design and Simulation Division, CSIR-Central Mechanical Engineering Research Institute, Durgapur 713209, India
[b] Department of Mechanical Engineering, Jadavpur University, Kolkata 700032, India

ARTICLE INFO

Article history:
Received 24 April 2013
Received in revised form 24 October 2013
Accepted 7 June 2014
Available online 20 June 2014

Keywords:
Pressure regulating and shut-off valve
Spool opening
Spool movement
Force balance
Dynamic simulation

ABSTRACT

Dynamic modeling of flow process inside a pressure regulating and shut-off valve has been investigated using a computational fluid dynamic approach. The valve is designed to reduce high inlet pressure to a lower level of outlet pressure which remains almost constant. With the change of inlet pressure, the change in position of the spool inside the valve was calculated using a force balance approach. The Navier–Stokes equation along with appropriate turbulent closure has been solved for this purpose in the compressible flow regime using ANSYS-FLUENT software with special functions developed for calculation of flow force. The code could predict the spool movement and the final spool position when the spool position is deviated from equilibrium. The final spool position and time required to reach equilibrium, besides the flow parameters, also depends on the value of friction coefficient between spool and the valve-body. Higher values of friction coefficient between the spool and the vale body is found to be associated with faster stability of the spool.

1. Introduction

Pressure regulating valve is an important component in many fluid power systems especially in aircraft systems. The valve is designed to reduce high inlet pressure, which may vary within a wide range, to a lower level of outlet pressure which remains almost constant. With the change of inlet pressure, position of the spool inside the valve also changes so that outlet pressure remains constant. Valve must be designed suitably so that spool chattering does not arise. Computational fluid dynamics (CFD) can play an important role in analyzing flow processes inside a valve providing important insight.

In recent years, CFD is increasingly being used to supplement experiments for product development. The numerical simulations reduce product development time as one can perform parametric studies without building prototype. Some notable works on valve CFD would be briefly mentioned here.

Among major works in the area of interest, Amirante et al. [1] have modeled commercially available direct proportional valve using CFD for evaluating the global performance. The study could show that compensation techniques based on spool profiling effectively balances the flow force at different level of valve openings. Methodology to evaluate flow forces on an open center direction control valve was again discussed by Amirante et al. in [2,3]. The studies do not report the detailed turbulent structure. Analytical treatment of closed center direction control valve has been attempted by Borghi et al. [4]. Fluid–structure interaction in valve spool under induced jam fault has been investigated by Deng et al. [5] which included the effect of viscous heating which is otherwise very difficult to investigate experimentally. Critical information such as estimating the flow coefficient can be achieved by solution of the flow field. Using a quasi-3d, i.e. an axi-symmetric formulation. Szente and Vad [6] estimated flow coefficients for small scale pneumatic solenoid valves. In another study of Szente and Vad [7], the effect of supersonic condition on the flow coefficient and the valve seat angles are summarized through a correlation. Some notable work on valve CFD are reported in studies e.g. flow visualization in a homogenizing valve by Stevenson et al. [8], modeling of four way direction control valve [9], and on butterfly valve [10] and ball valve by Chen et al. [11]. Chattopadhyay et al. [12] have investigated turbulent flow structure inside a PSROV using ANSYS-FLUENT. They have discussed the critical issues of handling compressibility and turbulence. In this work, both 2-D and 3-D simulations were performed with a conclusion that 2-D model could predict the flow coefficients satisfactorily. However, three dimensional analyses is required when the turbulent structure is to be resolved in detail.

Though dynamic modeling of valve is reported as early as in 70s [13], relatively fewer studies are found addressing valve movement using CFD approaches. Song et al. [14] have reported 2-D dynamic analysis of a pressure relief valve using CFX package.

* Corresponding author. Tel.: +91 9 33 2151376; fax: +91 9 33 23357254.
E-mail address: chimadri@gmail.com (H. Chattopadhyay).

http://dx.doi.org/10.1016/j.compfluid.2014.06.011
0045-7930/

Srikanth and Bhasker [15] studied the compressible air flow in a typical puffer chamber with moving contact between fixed electrodes using moving mesh technique and CFX solver.

In the present work, dynamic spool movement inside a pressure regulating valve has been investigated through computational flow analysis using a commercial package ANSYS-FLEUNT [16].

The valve under study is a solenoid operated in line sleeve valve as shown in Fig. 1. As per the initial design a 3D CAD model, shown in Fig. 1(a), has been made using UG NX commercial software. CAD model in Fig. 1(b) shows an inside view of the same pressure regulating valve. This valve consists of mainly three parts, valve body, spool and pilot chamber pressure control circuit as shown in the schematic diagram in Fig. 1(c). The valve is installed in a pipe line in an air flow path where available inlet pressure to the valve may vary within a wide range. The sole function of the valve is to regulate the air flow and keep the outlet pressure within the band as shown in Fig. 1(d). Its application is envisaged in bleed air regulating system of aircraft. Pressure control is achieved by energizing the solenoid valve. Air from upstream pressure is taken to the pilot chamber pressure control circuit through an orifice. It regulates the pressure applied to the pilot chamber side of the spool depending on the relief valve setting. On the other side of the spool, downstream pressure acts and causes the spool to move in appropriate direction.

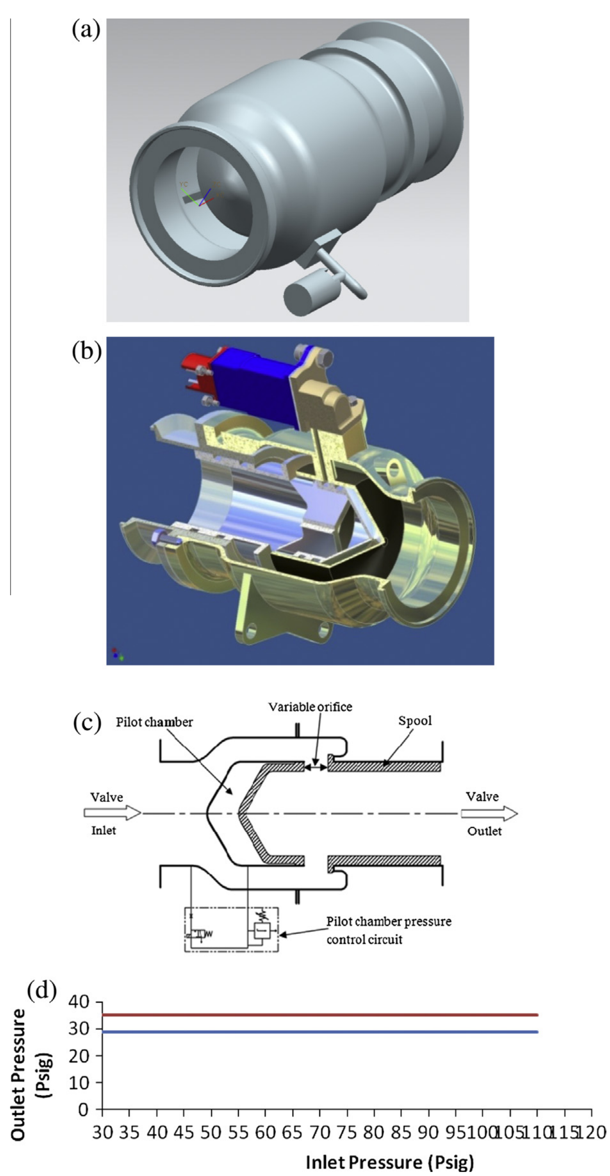

Fig. 1. (a) Typical solid model, (b) an inside view and (c) schematic diagram of a PRSOV valve, (d) outlet pressure band for different input pressure.

2. Mathematical modeling of flow

The fluid flow in the present investigation is in the compressible regime. Hence, compressible flows as described by the standard continuity and momentum equations are solved by FLUENT; however density of the flow has to be computed by Perfect Gas Law.

Unsteady, compressible and turbulent flow equations derived from the conservation laws of mass, momentum and energy are solved numerically in the commercial software package ANSYS FLUENT.

The governing equations to be solved are as given below:

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \bar{v}) = 0 \tag{1}$$

Momentum equation:

$$\frac{\partial}{\partial t}(\rho \bar{v}) + \nabla \cdot (\rho \bar{v} \bar{v}) = -\nabla p + \rho(v_l + v_t)\nabla^2 \bar{v} + \rho \bar{g} \tag{2}$$

The terms v_l and v_t refer to molecular diffusivity (kinematic viscosity) and turbulent diffusivity, respectively.

Energy equation:

$$\frac{\partial T}{\partial t} + \bar{v}\nabla T = (\alpha + \alpha_t)\nabla^2 T + \nabla v \bar{\tau} \tag{3}$$

Here α and α_t refer to thermal diffusivity and corresponding turbulent diffusivity, respectively. The last term in the energy equation represents viscous dissipation which was duly considered to arise out of viscous stress τ.

The turbulent viscosity term v_t is to be computed from an appropriate turbulence model. In this approach, the turbulent viscosity is computed using two different equations for parameters such as turbulent kinetic energy k and dissipation rate ε. In this study, for closing the time-averaged momentum equation, a realizable k–ε model, proposed by Shih et al. [17] was chosen. As such, the model was found to work satisfactorily as observed by Chattopadhyay et al. [12]. The expression for the turbulent viscosity is given as:

$$\upsilon_t = C_\mu \frac{k^2}{\varepsilon} \tag{4}$$

The turbulent diffusivity is related to the molecular diffusivity in the following manner

$$\mathrm{Pr}\frac{v_t}{\alpha_t} = \mathrm{Pr}_t \tag{5}$$

The value of turbulent Prandtl number Pr_t is chosen as 0.85. The detailed description of turbulent model is outlined in Chattopadhyay et al. [12].

The working fluid is air albeit in the compressible regime with varying density which has been modeled using Perfect Gas Law. The viscosity has been modeled using Sutherlands viscosity for air which can be expressed with three coefficients as:

$$\mu = \mu_o \left(\frac{T}{T_o}\right)^{3/2} \frac{T_0 + S}{T + S}$$

where μ_0 and T_0 are reference viscosity and temperature and S is Sutherland constant.

A value of $\mu_o = 1.716 \times 10^{-5}$ kg/m.s, $T_0 = 273.11$ K and $S = 110.56$ K were used in the present work.