Computational Fluid Dynamics Investigations of Shock Waves in Safety Relief Valves

Wael Elmayyah*

4th April 2013

Abstract

Safety Relief Valves plays an important role in any pressurised system. The proper design of safety relief valves should consider the effective opening and closing characteristics to achieve the desired valve performance. One of the challenges in any valve design is the elimination of Shock waves. Shock waves are characterised by sudden pressure increase and Mach number drop.

Detecting shock waves in safety relief valves using computational fluid dynamics could asses valve design to eliminate the shock waves. I t have been proven that the Reynolds Averaged Navier Stocks (RANS) equations with the k-ϵ turbulence model could model the internal gas flow in safety relief valves. The RANS and k-ϵ turbulence have been solved by a commercial CFD code. In safety valves discharge flow rate and closing and opening behaviours depend on the upstream pressure and certain critical areas. In this study, the effect of the upstream pressure has been discussed. Upstream pressure range was from 7 to 140 bars. The effect of the valve geometry has been discussed as well. Different critical outlet areas and divergent angles have been changed to investigate its effect on the shock wave parameters.

The upstream pressure effect on shock wave occurrence and intensity have been studied. The results showed that, computational fluid dynamics can be used to predict the location of the shock wave. In addition to determining the pressure peak and the Mach number drop and other properties which help eliminating shock wave occurrence in valve design. An efficient design should consider the geometrical and flow parameters that affect the shock wave occurrence and intensity.

Keywords:

Computational Fluid dynamics, Shock Wave, Reynolds Averaged Navier Stocks (RANS), Safety Relief Valves.

*Military Technical College, Egyptian Armed Forces

Nomenclature

M	Mach Number (-)
p	Pressure (pa)
γ	specific heat ratio (-)

1 Introduction

1.1 Shock waves in Safety Relief Valves

Where there are supersonic flows, there are usually also shock waves. A fundamental type of shock wave is the normal shock wave – the shock wave normal to the flow direction. If the speed of the flow is much less than the speed of sound in the gas, the density of the gas remains constant and the flow of gas can be described by conserving momentum, and energy. As the speed of the flow approaches the speed of sound, compressibility effects on the gas must be considered. The density of the gas varies locally as the gas is compressed by the interior geometry. For compressible flows with little or small flow turning, the flow process is reversible and the entropy is constant. The change in flow properties are then given by the isentropic relations. But when a flow moves faster than the speed of sound, and there is an abrupt change in the flow area, shock waves are generated. Shock waves are very small regions in the gas where the gas properties change by a large amount. Across a shock wave, the static pressure, temperature, and gas density increases almost instantaneously. The changes in the flow properties are irreversible and the entropy of the entire system increases. Because a shock wave does no work, and there is no heat addition, the total enthalpy and the total temperature are constant. But because the flow is non-isentropic, the total pressure downstream of the shock is always less than the total pressure upstream of the shock. There is a loss of total pressure associated with a shock wave. Because total pressure changes across the shock, the usual (incompressible) form of Bernoulli's equation across the shock can not be used. The Mach number and speed of the flow also decrease across a shock wave.[7]

A common example where shock waves appear, is a supersonic nozzle flow, which is typically found in a jet or rocket engine. A normal shock can appear in the diverging part of the nozzle under certain conditions. The nozzle geometry is very close to the safety valve geometry and hence the shock wave occurrence is possible. Relatively small valve geometry and hence the high speed flow through safety relief valves encourage the formation of shock waves in safety relief valves. Shock waves result in noise and vibration and may cause system failure. The designed valve's geometry and operational conditions should limit or eliminate the formation of shock waves. Therefore, many studies have been carried out to understand, analyse and predict

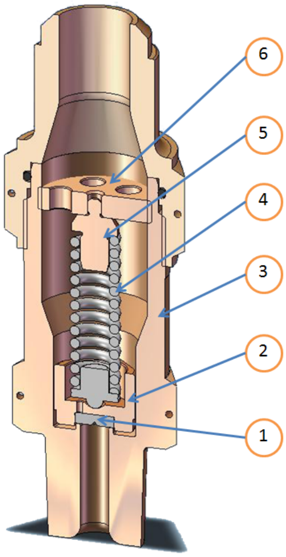

Figure 1: Safety Relief Valve Construction

shock waves in external and internal flow through pipes, nozzles and diffusers.[6][5]

1.2 Valve construction and operation

A conventional spring loaded safety relief valve is considered in this study. It is a 1/4" inlet size. Figure 1 shows a cross sectional drawing of the entire valve. The safety relief valve is set to open at a pressure safely below the bursting pressure of a pressurised system. The piston (2) is held against the seat (1) by a loaded spring (7) which is fitted between the gland insert (5) and the spring guide (6); excessive pressure forces the piston to open. The valve is designed such that when the valve opens slightly, the pressure builds up to open it fully and to hold the valve open until the pressure drops a predetermined amount. The relieving pressure is set by the initial compression of the spring which can be altered through the adjusting gland (4). Minor modifications have been applied to the valve to facilitate the experimental work; a 4 mm diameter rod 150 mm long has been fitted to the piston rear, the adjusting gland (4) and the adjusting gland insert (5) have been combined and replaced by a single equivalent component and the spring (7) has been cut to a smaller length. The rod diameter is less than the spring inner diameter, so there is no significant change in the flow area at this region. However, the rod has been chosen to have the minimum diameter that could resist the aerodynamic forces without buckling or failure. Tests on similar valves indicate that these modifications have no significant effect on the measured or the predicted values of the mass flow rate, back pressure or the fluid forces on the piston Dempster et al [2][3] [1]

2 Mathematical Work

2.1 Computational Work

To allow computational efficiencies a two dimensional axisymmetric model has been developed to represent the safety valve geometry. A two dimensional axisymmetric model has been shown to provide adequate prediction for the mass flow rate and piston force in previous research work Dempster et al [2]. for similar safety valve geometry and flow conditions. In this model the flow areas between the piston and the body and the gland exit holes, shown on Figure [Flo:mesh grid half-1], have been represented as equivalent annulus areas. The flow area around the piston is a very important area for predicting the air flow rate, hence it has been maintained when modelled as an annulus flow area. The piston front face area is also required to be maintained for predicting the piston force. Therefore, the piston seat area has been kept the same (since it is already symmetric), whereas the piston front face outer diameter has been chosen to keep the same piston front face area and hence the flow area around the piston. The gland flow area (5 holes) has been modelled as an annulus area the centre of which is located at a radius of half the valve body radius. The computational mesh has a total of 7000 quadrilateral cells distributed giving an average mesh density of 7 $cells/mm^2$. A more dense mesh of 14000 quadrilateral cells has been used to examine the grid independency, with no significant improvement for the solution. The difference in air flow rate was 0.00001 kg/s and 150 Pa for the back pressure so the cell number was kept about 7000 in all cases.

2.1.1 Boundary Conditions and Solution

The proper setting of the boundary conditions is an essential step to obtain accurate CFD results. The boundary conditions are applied at the valve entrance, valve outlet and valve walls. Valve walls were defined as stationary walls. At the inlet boundary, the stagnation pressure, static pressure and stagnation temperature are applied; in addition an initial value for the turbulence intensity and the hydraulic diameter are introduced. Hence, an initial air mass flow rate is determined at the inlet area then is recalculated from the downstream conditions at the choking plane. At the outlet boundary conditions the static pressure and the stagnation temperature are applied. However, the flow calculation is independent of the outlet boundary condition since the flow is choked for all test pressures in this study. The discertization scheme used for the continuity, momentum, energy, turbulent kinetic and turbulent dissipation energy equations was second order upwind for the convection terms and second order central difference for the diffusion terms. The convergence criterion was based on the residual values of the calculated variables, i.e., mass, velocity components, energy, turbulent kinetic energy and turbulent dissipation energy. The threshold values were absolute with magnitudes of 1×10^{-3} for all variables, with the exception of the energy equation where it was 1×10^{-6}. The pressures range used was 7 - 140 barg (100 - 2000 psig).

The computational model has been validated by comparing the predicted results by the experimental results carried out by Elmayyah and Dempster [4]. Due to the limitation of the experimental facilities shown in [4], the comparison has been limited to maximum of 15 barg.Figure 2 shows the comparison of the predicted and measured values for the mass flow rate and the pressure distribution. The results show a good agreement.

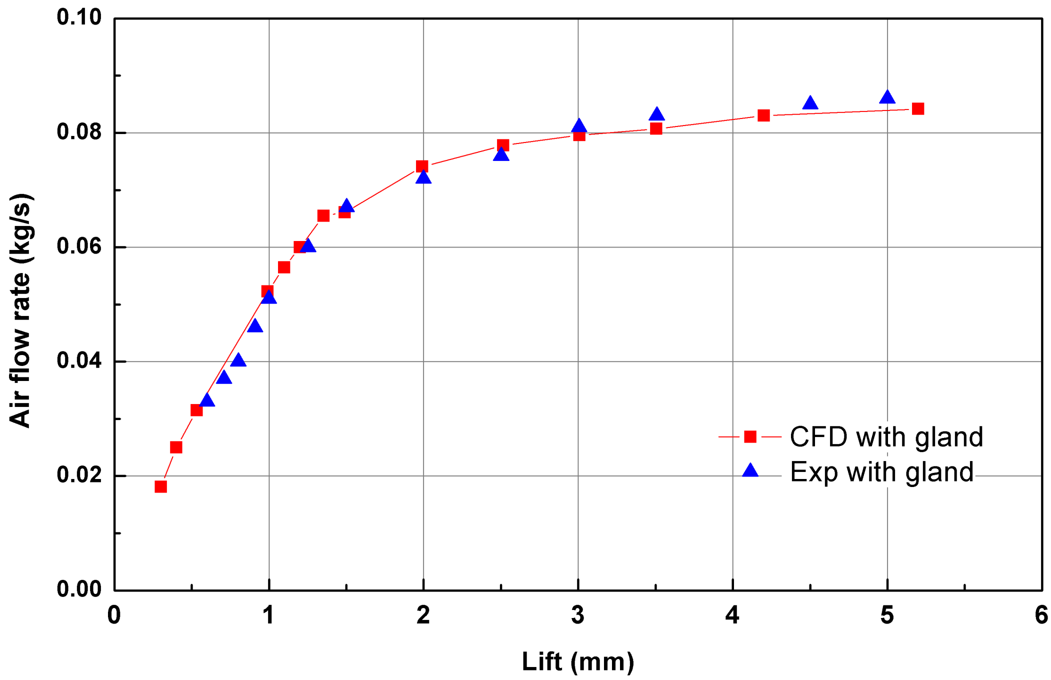

Figure 2: The predicted and measured Air flow rate

2.2 Simplified Models

Another way to check the accuracy of the Computational model is to compare the results with the simplified models that can predict the pressure, temperature, density and the Mach number downstream the shock wave. This requires determining the shock wave location, which is not an easy job. The pressure and Mach number upstream the shock wave have been determined by the computational fluid dynamics. Thereafter, the downstream pressure and Mach number have been predicted using the simplified models. The predicted values by the simplified model and the results predicted by the CFD.

$$M1^2 = [((\gamma - 1)\, M^2) + 2]/[2\gamma M^2 - (\gamma - 1)] \tag{1}$$

$$p1/p0 = [2\gamma M^2 - (\gamma - 1)]/(\gamma + 1) \tag{2}$$

Where

M Mach number upstream the shock wave. (-)

$M1$ Mach number downstream the shock wave. (-)

$\frac{p1}{p0}$ Ratio of pressure upstrem and downstream the shock wave. (-)

γ Heat specific ratio of air. (-)

3 Results and Discussion

The pressure and Mach number distribution through the safety valve can predict the shock wave occurrence and determine its location and intensity. Figures 3 and 4 present the pressure distribution on the valve side at the location of the shock wave with and without shock wave respectively. From investigating the pressure distribution, it could be noticed that the shock wave take place between low pressure and high pressure areas (marked by the dotted line in Figure 4). Figures 5 and 6 illustrate the pressure and Mach number profile at the valve side where the shock wave location could be noticed. The pressure increase and the Mach number drop at the shock wave location is presented in Figures 7 and 8 respectively.

It is clear from Figure 8 that the Mach number profile doesn't depend on the Inlet pressure. This is because of the restricted chocked flow at the critical chocking plane as discussed by and Dempster and Elmayyah [1]. At fixed chocking plane the Mach number is unity independent of the inlet pressure value. Therefore, the Mach number profile doesn't change with the inlet pressure value.

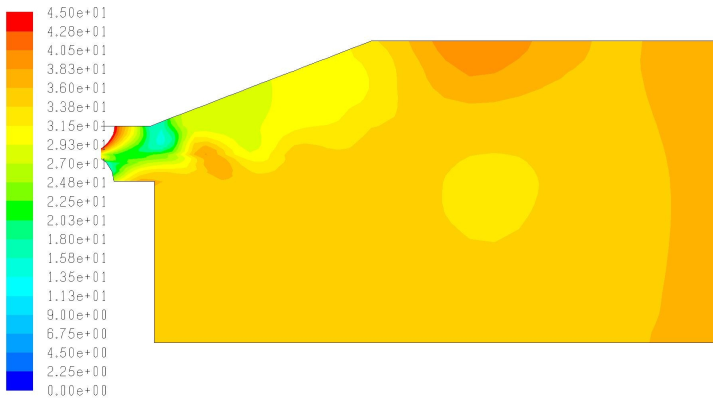

Figure 3: Pressure Distribution with no Shock Wave

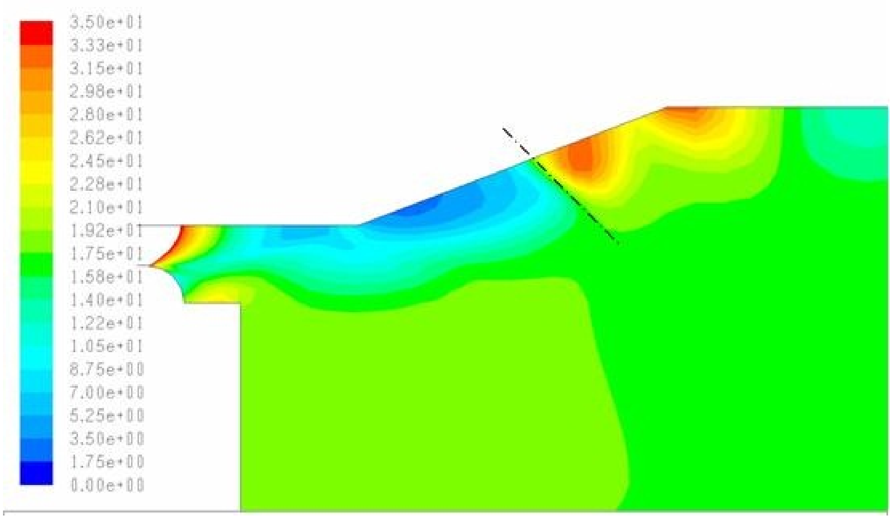

Figure 4: Pressure Distribution at Shock Wave Location

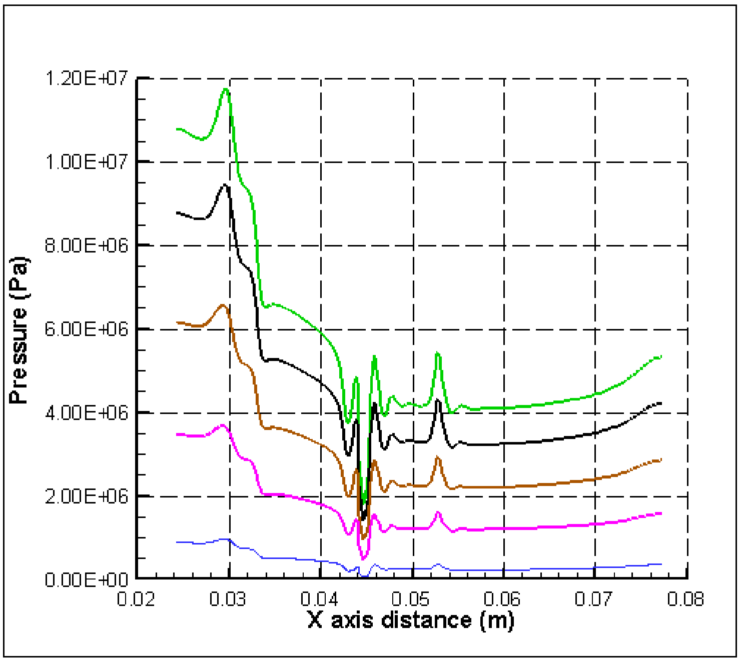

Figure 5: Pressure profile at valve side

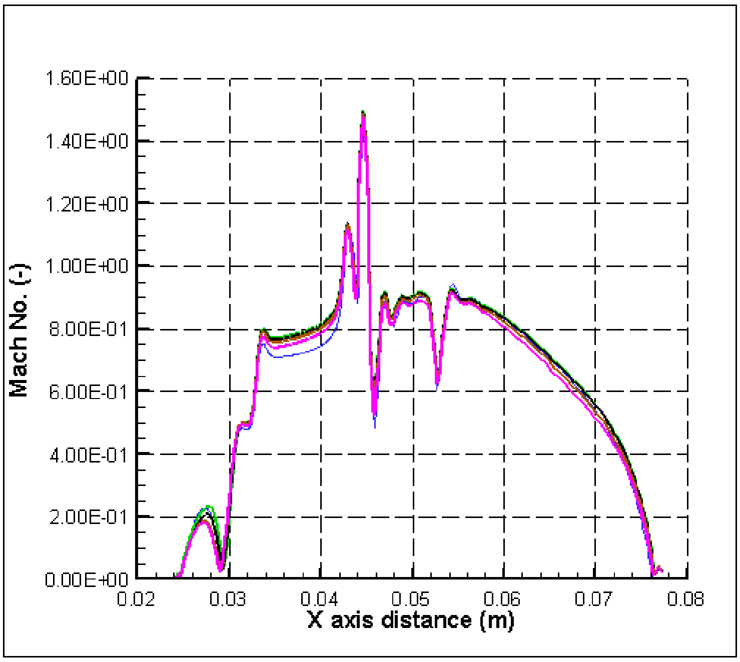

Figure 6: Mach number profile at valve side

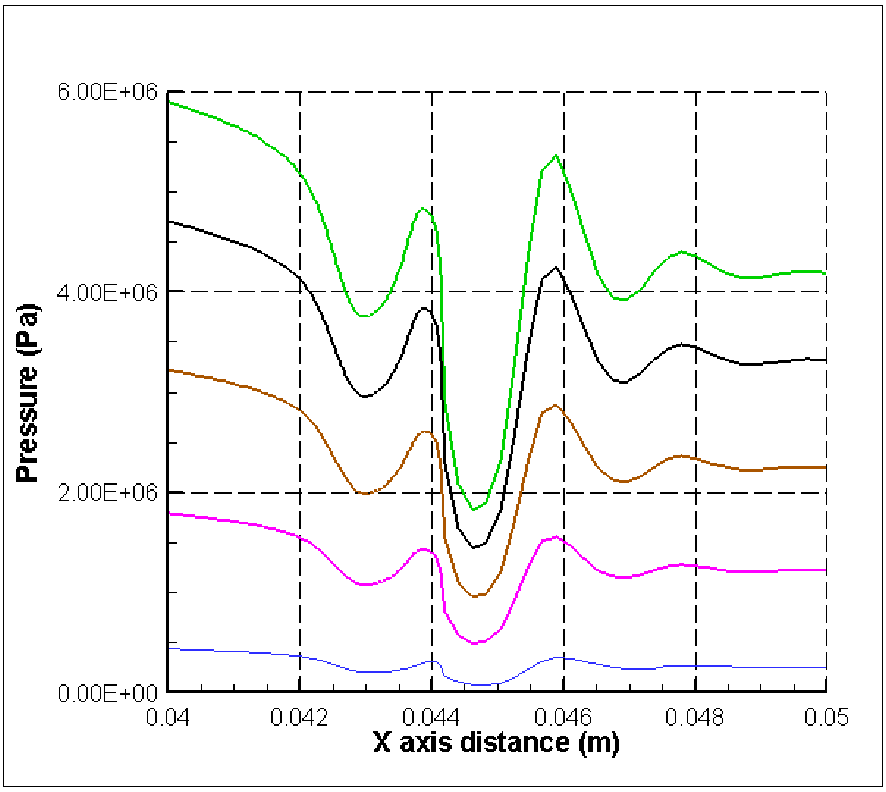

Figure 7: Pressure profile at shock wave location

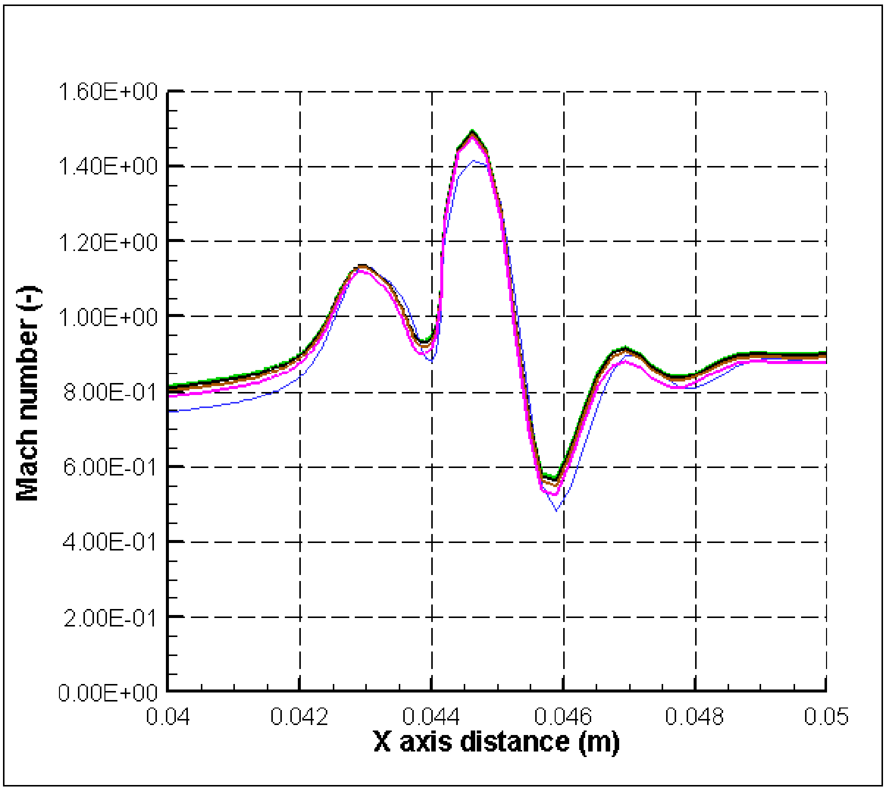

Figure 8: Mach number profile at shock wave location

Equations 1 and 2 have been used to calculate the downstream static pressure and Mach number at different pressure values from figure 7. The calculated results have been compared with CFD predicted results in Figure 3. Good agreement between both results has been shown which encourages using CFD in predicting shock wave intensity.

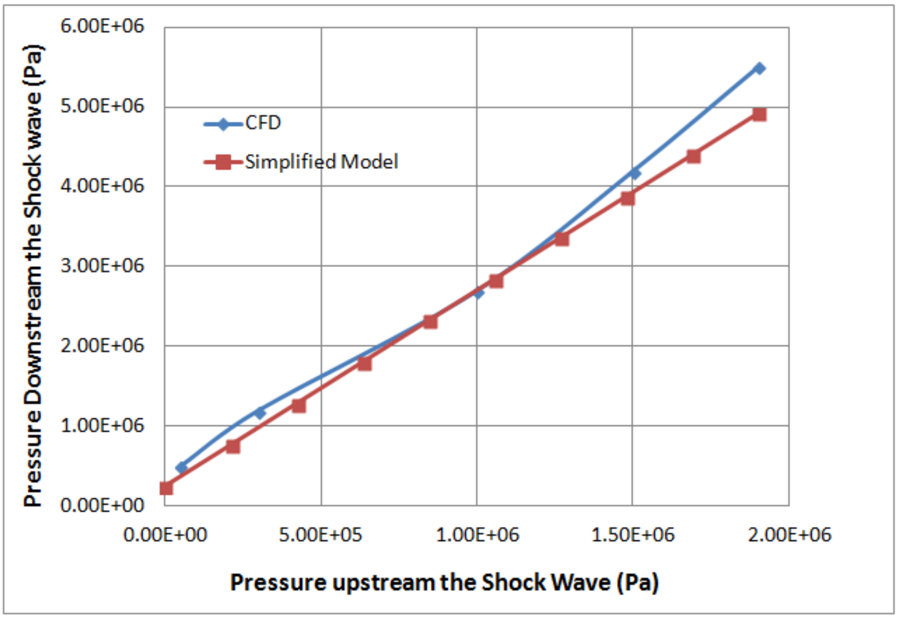

Figure 9: CFD predicted and calculated Static Pressure downstream the shock wave at upstream Mach number M=1.5

4 Conclusions

Computational Fluid Dynamics can be used in predicting shock waves in internal flow. A normal Shock wave has been detected at a safety relief valve. Pressure increase and Mach number have been determined. The pressure increase across the shock wave is significantly dependent on the upstream pressure. On the other hand, The Mach number drop is independent of the stagnation pressure due to the fixed chock plane with the value Ma=1. This makes the Mach number profile is constant at any value of stagnation pressure.Predicting Shock waves helps eliminating its occurrence in valve design. An efficient design should consider the geometrical and flow parameters that affect the shock wave occurrence and intensity.

References

[1] W. Dempster and W. Elmayyah. A computational fluid dynamics evaluation of a pneumatic safety relief valve. The 13th International conference on Applied Mechanics and Mechanical Engineering (AMME), May 2008.

[2] W. Dempster, C. K. Lee, and J. Deans. Prediction of the Flow and Force Characteristics of Safety Relief Valves. *Proceedings of PVP2006-ICPVT-11 2006 ASME Pressure Vessels and Piping Division Conference*, July 2006.

[3] William Dempster and Wael Elmayyah. Two phase discharge flow prediction in safety valves. In *13th International Conference on Pressure Vessel Technology*, May 2012.

[4] Wael Elmayyah and William Dempster. Prediction of two-phase flow through a safety relief valve. *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering*, August 2012.

[5] A. F. P. Houwing, K. Takayama, Z. Jiang, T. Hashimoto, K. Koremoto, H. Mitobe, and M. J. Gaston. Abel inversion of axially-symmetric shock wave flows. *Shock Waves*, 14(1):21–28, June 2005.

[6] Doyle Knight, Hong Yan, Argyris G. Panaras, and Alexander Zheltovodov. Advances in CFD prediction of shock wave turbulent boundary layer interactions. *Progress in Aerospace Sciences*, 39(2-3):121–184, February 2003.

[7] Koji Miyaji and Kozo Fujii. Simulations of unsteady shock wave reflections using adaptive unstructured grids Fifteenth International Conference on Numerical Methods in Fluid Dynamics. In Paul Kutler, Jolen Flores, and Jean-Jacques Chattot, editors, *Fifteenth International Conference on Numerical Methods in Fluid Dynamics*, volume 490 of *Lecture Notes in Physics*, chapter 50, pages 334–339. Springer Berlin / Heidelberg, 1997.