Numerical Study on the Flow Characteristics of a Solenoid Valve for Industrial Applications

TAEWOO KIM [1], SULMIN YANG[2], SANGMO KANG [3]
[1,2,4] Mechanical Engineering
Dong-A University
840 Hadan 2 Dong, Saha-Gu, Busan-604-714
SOUTH KOREA
[1]donenom@nate.com [1]http://cfdlab.donga.ac.kr

Abstract: - Study on the flow characteristics of solenoid valves is essential since they play pivot role in a flow circuit for controlling the flow through a required path for a specified duration. The flow characteristics of a 3/2 -way solenoid valve used in the four stroke propulsion engine of a ship is studied by performing numerical simulation using ANSYS CFX 12.0 commercial software package. The solenoid valve can be operated in two ways- either by pushing the hand operated button for moving the spool or by moving the spool using electric signal. The valve under study is a superspeed valve and the response time of moving spool is very short. The movement of the spool which is located in the middle of the body of solenoid valve affects the flow characteristics. This is examined by performing a static analysis by changing the position of the spool. The high pressure air passes through a pipe and reaches the engine tank of given dimensions. The simulation is performed for different inlet pressure and spool displacement and the corresponding flow characteristics are obtained. The results of the three dimensional numerical simulation will be helpful in the proper design of solenoid valve for the industrial applications.

Key-Words: -Numerical simulation, Computational fluid dynamics, Solenoid valve, Spool displacement, Flow characteristics, ANSYS CFX, SST model.

1 Introduction

Shipbuilding industry is the equipment industry where the various fields of engineering like mechanical, electrical and electronics, industrial engineering, chemical engineering, etc are involved. This industry plays a significant role in the growth of national economy. The demand for the production of marine engines which are the key component of ship are increasing in the present industrial era. The shipbuilding equipment industry has seen a rapid growth in the recent times in most of the nations. But manufacturing industry that produces propulsion engine for the ship has facing the problems of necessary engineering analysis both quantitatively and qualitatively for the better design and utilization of different components of the engine.

Here, we performed a numerical study on the fluid flow characteristics of an industrial 3/2 way solenoid valve which is a pivot element in the propulsion engine of a ship. Computational fluid dynamics (CFD) is used as the numerical tool for the present simulation study.

In a typical engine (Four-stroke propulsion engine) the 3/2 way pneumatic valve is located on the engine. Commonly two 3/2 way pneumatic valves are placed in a row. One of the valves work as a starting device for generating pneumatic pilot pressure (30bar) during starting of the engine. The other valve also generates pressure which is used for the emergency stopping of the engine by cutting the fuel supply.

Compared to engine systems which has distributors for generating the pressure for starting the valve, the 3/2 way pneumatic valve has a main feature that it removes the distributors which generates the specific pressure, because the solenoid valve itself acts for controlling the pressure. For this reason, 3/2 way valves are applied to most of the electronic control engines, and the demand for these valves are increasing gradually.

The valve under study is a superspeed solenoid valve for which the reaction time of the spool is very short. In industrial applications, this valve transports the high-pressure air to the engine. Therefore, we are interested to analyze how to transport effectively the high-pressure air from the inlet hole of the valve to the engine passing through the spool part.

Fig. 1 represents the sectional view of the 3/2 way solenoid valve. The upper part of the push button is the solenoid part. And the one which is located in the body is the spool that can be moved up and down. In industrial applications, the high pressure air is maintained to move the spool upwards. When then spool moves down, the high-pressure air flows through the gap between spool and the plane. In very short time, the spool undergoes maximum displacement and return to its original position and restores the sum of spring force and vertical force from the bottom.

The solenoid valve can be operated in two ways- either by pushing the hand operated button for moving the spool or by moving the spool using electric signal. A typical solenoid valve has two parts- the solenoid part and the body part. The body part of the valve under study has one inlet hole and two outlet holes.

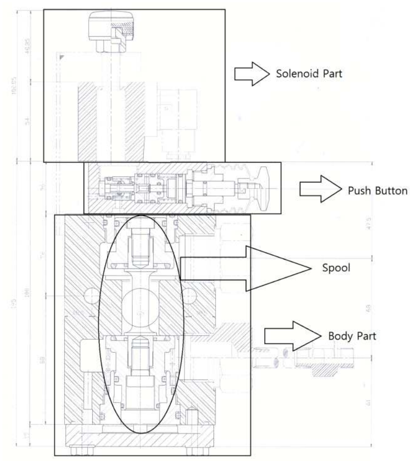

Fig. 1 Schematic diagram of 3/2 way solenoid valve

The diameter of inlet hole and the two outlet holes of the valve is about 31.78mm, so we choose it to be 32mm. The distance from the pipe to the inlet hole is 50mm, and there are two 150mm long pipes which are connected to a tank of dimensions 88X88X88 mm^3 from the two outlet holes. The maximum displacement of the spool which is located in the body part is 7.5mm. Hence, we performed the simulation for various cases of spool displacement from the contact surface-1.5mm, 3mm, 4.5mm, 6mm, and 7.5mm, and each case has analyzed separately for different working pressures of 10bar, 20bar and 30bar. The numerical simulation is done using computational fluid dynamics as the numerical tool. Computational fluid dynamics (CFD) has proved its capability in the mainstream scientific research and in the industrial engineering communities.

Recently, CFD simulation using commercial software has become popular for studying fluid flow characteristics of mechanical equipments. We used ANSYS CFX 12.0 which is a commercial CFD tool for fluid flow analysis. The flow characteristics of the valve depends significantly on the fluid properties and time hence we conducted transient analysis for unsteady condition.

2 Problem Formulation

2.1 3D Modeling of solenoid valve

For the numerical analysis of fluid flow and pressure field, the 3D model of 3/2way solenoid valve is created using SolidWorks at a 1:1 scale. Fig.2 shows the 3D model.

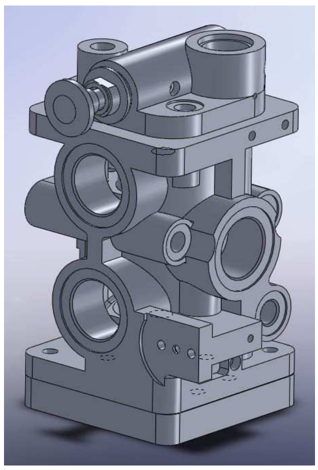

Fig. 2 3D Model

2.2 CFX Mesh for the fluid field

Fig.3-8 shows the different views of the inlet and outlet part of the solenoid valve along with the meshes created for the simulation. The meshes are generated using ANSYS CFX –Mesh. In all the above figures the spool is located 7.5mm away from the plane. The inlet part which maintain the high pressure air at the beginning is shown in Fig. 3-5.The right side part in Fig. 3-4 is the inflow part of high-pressure air from the air-compressor, and the left-top side of Fig. 4 is connected to the outlet fluid field. Fig. 6-8 shows the outlet fluid field which maintain atmospheric pressure at beginning. The left-bottom side of Fig. 8 is connected to the inlet fluid field.

The quality of meshing has significant effect on the accuracy of the simulation. Theoretically, the more elements in the geometry, the higher mesh quality, and the better is the accuracy of the results. But computational time will be more. Based on CFX user's manual and several simulations with various meshing size, the total number of nodes is 215,383 and the total number of elements is 975,786. Also we applied the meshes densely to the fluid field around the spool where the pressure changes are high.

Fig. 3 Inlet part of Solenoid valve
(Isometric view)

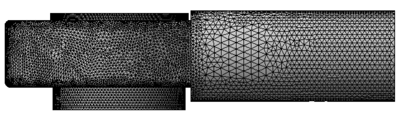

Fig. 4 Inlet part of Solenoid valve
(Side view)

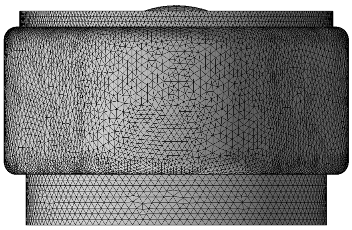

Fig. 5 Inlet part of Solenoid valve
(Front view)

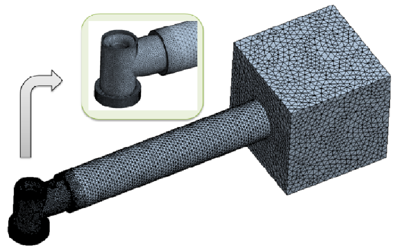

Fig. 6 Outlet part of Solenoid valve
(Isometric view)

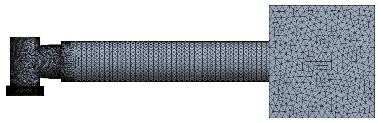

Fig. 7 Outlet part of Solenoid valve
(Side view)

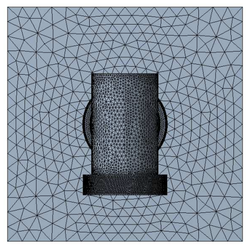

Fig. 8 Outlet part of Solenoid valve
(Front view)

2.3 Boundary Conditions

We applied the opening boundary condition, which means that the inflow and outflow of fluid is free, for maintaining high-pressure air to the inlet side of the valve. We performed the simulation for three cases with inlet side pressure of 10bar, 20bar and 30bar. The relative atmospheric pressure applied is taken as P=0.0 N/m^2. The temperature T is 300K(25℃). We assumed air as ideal gas and the flow is compressible since the working fluid is the high pressure air.

2.4 Shear Stress Transport Model

In the CFX simulation, we employed one of the most effective shear stress transport (SST) model. The model works by solving a turbulence/frequency-based model (k-ω) at the wall and k-ε in the bulk flow. A bending function ensures a smooth transition between the two models. This method has been found to be superior over the commonly used k-ω and k-ε model interms of accuracy.

3 Results and Discussion

The numerical simulation is performed by neglecting the pressure drop due to gravity and taking the flow as turbulent. The maximum displacement of spool in the body is 7.5mm. Also we conducted numerical study based on various spool displacements ranging from 0-7.5mm. We chose an operating pressure of 30 bar and we performed simulations for additional working pressures of 10, 20 and 30 bar.

The solenoid valve which is considered in this study has just one outlet pipe when the spool of body is fully moved (7.5mm). In this case, the entire stabilization of the pressure was completed in 0.02sec, and it was completed in 0.03sec when the spool is located in middle position.

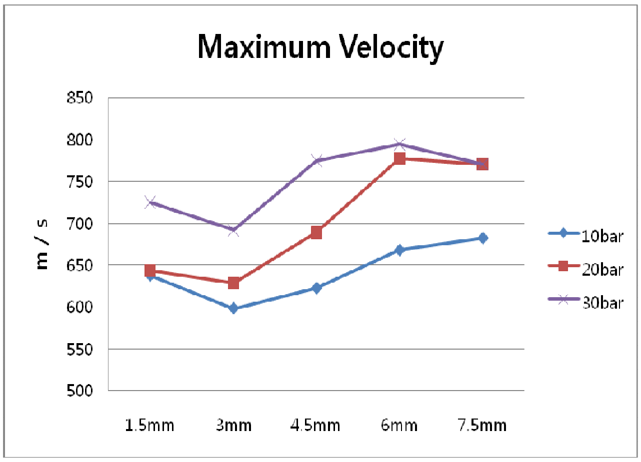

Fig. 9 Dependence of Maximum velocity on the spool displacement

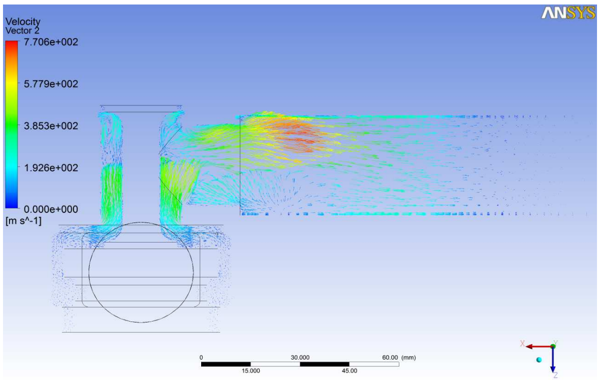

Fig. 10.a Local velocity field at 0.0002sec

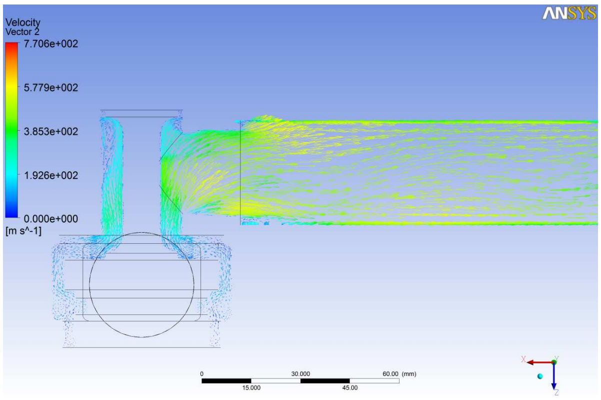

Fig. 10.b Local velocity field at 0.001sec

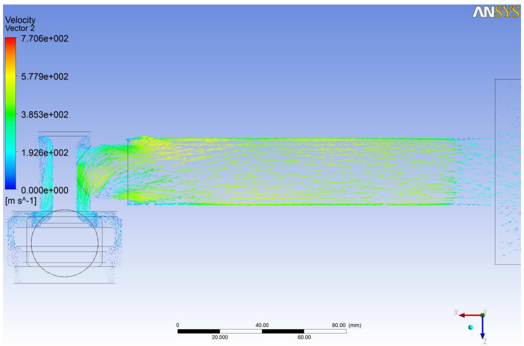

Fig. 10.c Local velocity field at 0.003sec

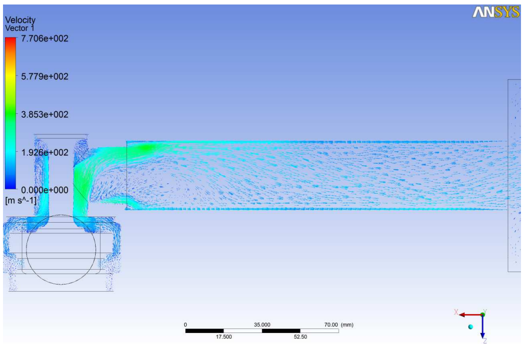

Fig. 10.d Local velocity field at 0.006sec

Fig.9 shows the variation of maximum velocity for different spool displacement. It can be seen that the velocity difference is largest for the opening pressure of 20bar.

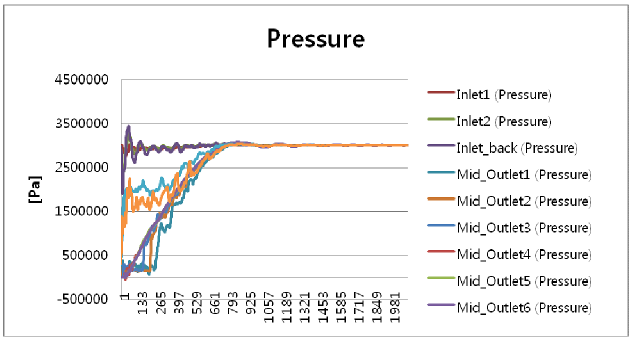

Fig. 11 Stabilization of pressure with respect to ime

Fig. 10 shows the visualization of velocity field at the outlet part of the valve. For 30 bar and 7.5mm spool displacement the velocity is maximum at 0.0002sec.The velocity decreases and then increases rapidly from 0.0004sec to 0.0008sec. At 0.001sec, many parts of the outlet have the velocity of about 580m/s, and then the velocity decreases from the tank wall at 0.003sec. At 0.006sec, the velocity decreases to 380m/s and then the velocity field is stabilized gradually.

Fig. 11 shows the pressure stabilization plot with respect to time for the fluid field around spool and inlet and outlet parts of the valve. Fig.12 depicts the variation of maximum pressure in the local area by the displacement of the spool. In the case of 10bar and 20bar, the variation of maximum pressure with respect to spool displacement is marginal. On the contrary, in the case of 25 and 30bar, the maximum pressure increases and decreases with increase of spool displacement .The maximum pressure suddenly increases when the spool undergoes a displacement of 3mm and 7.5mm.

Fig.13 represents the case when the spool is positioned at a distance of 7.5mm at an operating pressure of 30bar. The local pressure field is plotted for each 0.0002 sec time duration. Initially the high-pressure air goes up around the spool and high pressure is accumulated in the local area, and finally the entire pressure become equal with that of opening pressure for every 0.0002sec time interval.

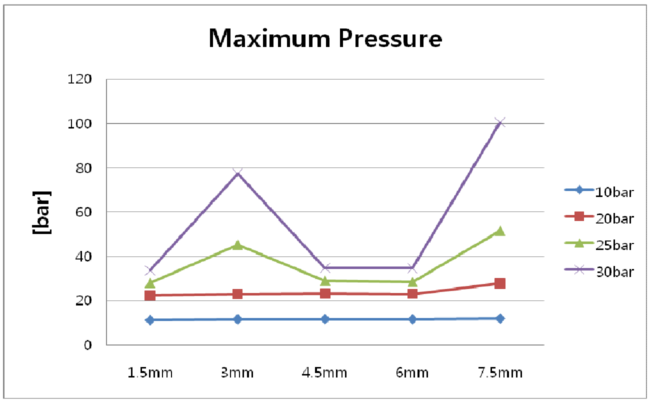

Fig. 12 Dependence of Maximum pressure on the spool displacement

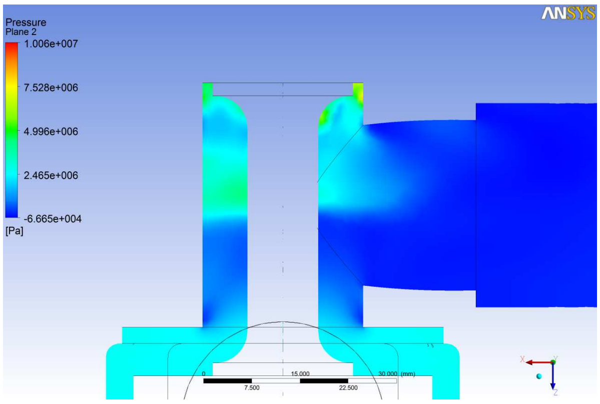

Fig.13.a Local pressure field at 0.0002Sec

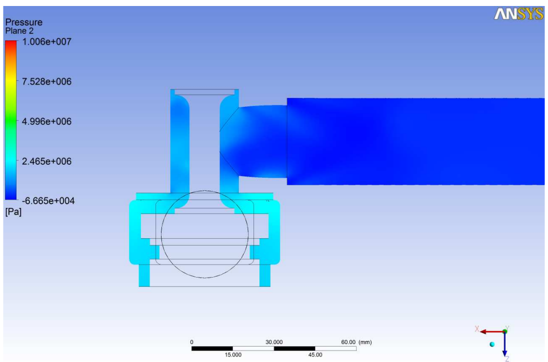

Fig.13.b Local pressure field at 0.0004Sec

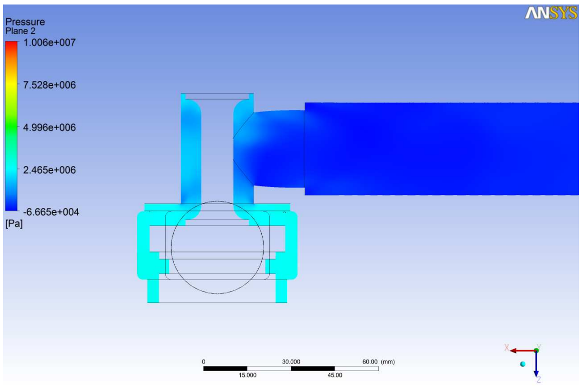

Fig.13.c Local pressure field at 0.0006Sec

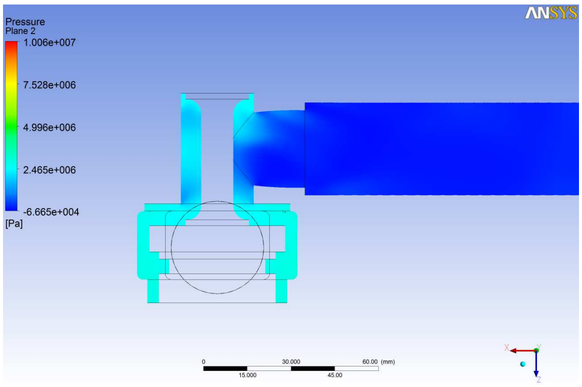

Fig.13.dLocal pressure field at 0.0008Sec

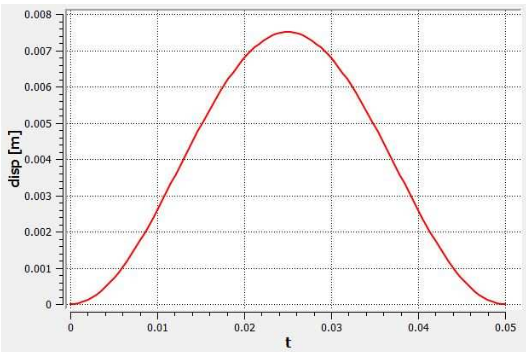

Fig. 14 Sin(t) Equation for the moving of the spool

In the CFX, we can insert the CEL(CFX Expression Language) for more detail setting of the domain. So, we draw the sin(t) graph using CEL for moving of the spool and observed the flow characteristics.

The total simulation time is 0.1[s] and the cycle timestep is 0.05[s].

$$S = 3.75[\mathrm{mm}]\times\sin((t/0.05[\mathrm{s}]\times 2\pi)+3\pi/2)+3.75[\mathrm{mm}]$$

Fig. 15 shows that Local Pressure field for the time interval.

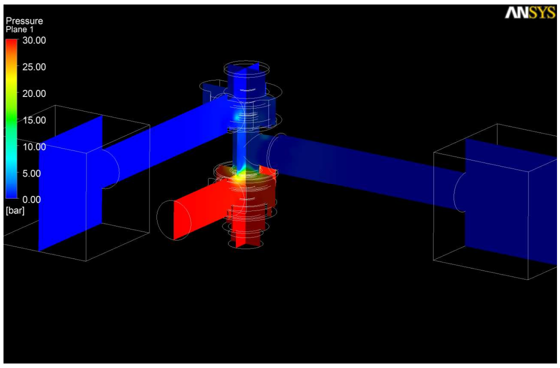

Fig. 15.a Local pressure field at 0.0002Sec

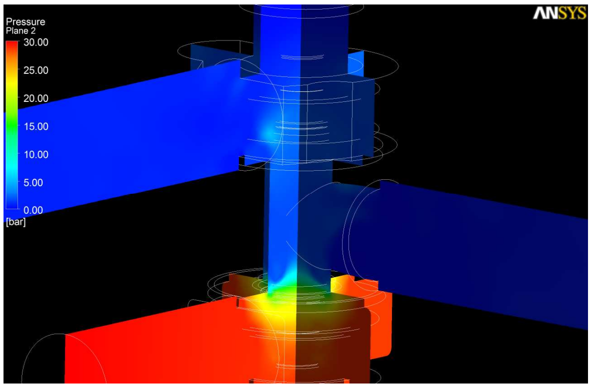

Fig. 15.b Local pressure field at 0.0006Sec

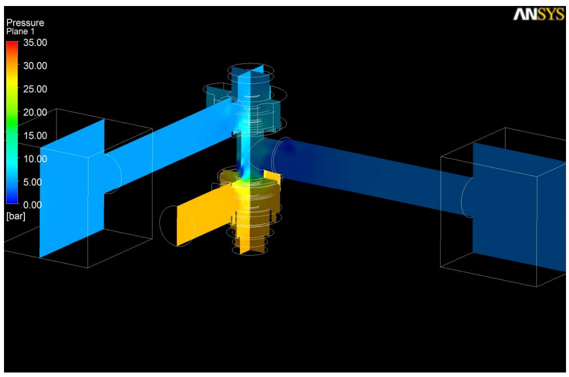

Fig. 15.c Local pressure field at 0.01Sec

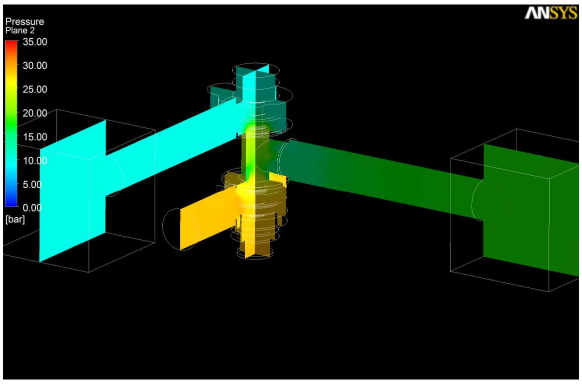

Fig. 15.d Local pressure field at 0.02Sec

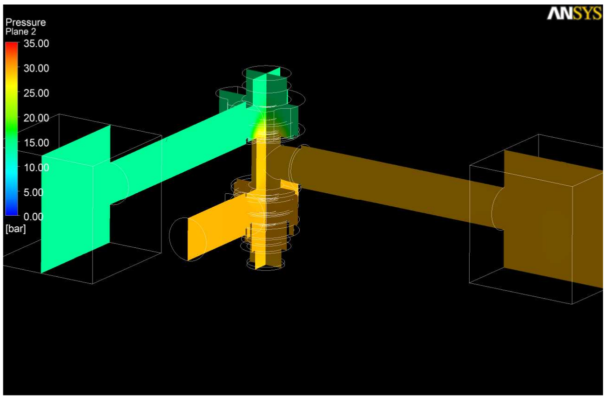

Fig. 15.e Local pressure field at 0.03Sec

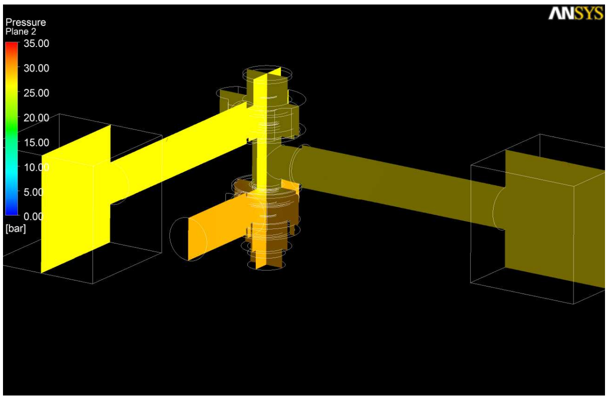

Fig. 15.f Local pressure field at 0.05Sec

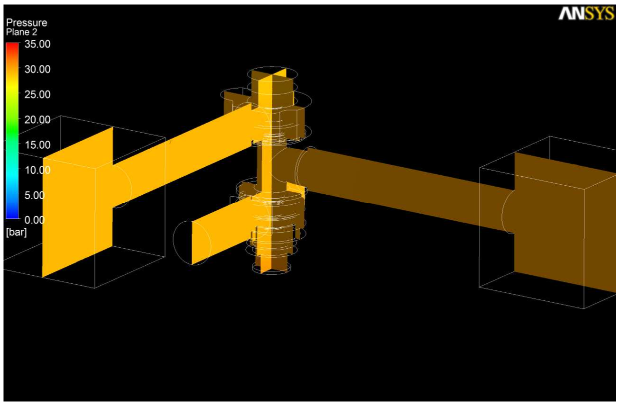

Fig. 15.g Local pressure field at 0.07Sec

In the half cycle, the middle pipe and tank is filled with high pressure air, and then the top pipe and tank is filled with high pressure air in rest cycle.

And the Fig. 16 shows that Local velocity field for the time interval.

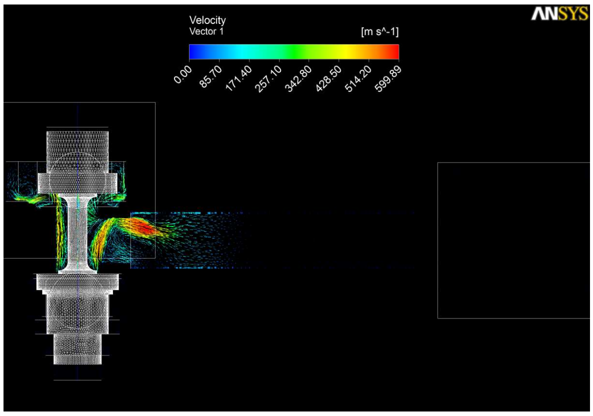

Fig. 16.a Local velocity field at 0.0002sec

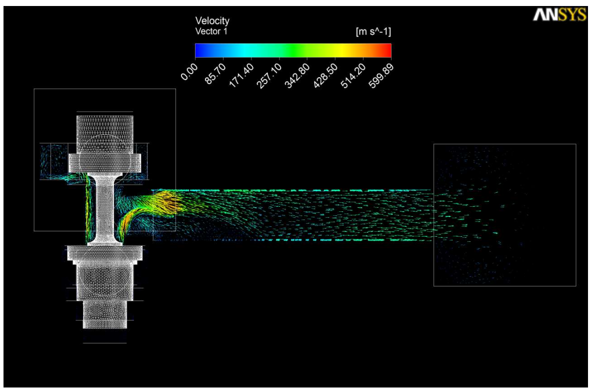

Fig. 16.b Local velocity field at 0.001sec

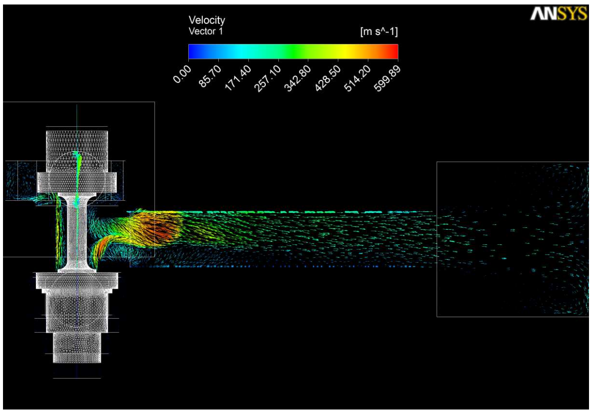

Fig. 16.c Local velocity field at 0.0018sec

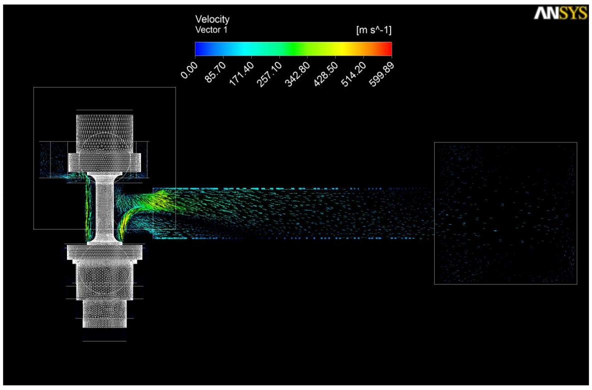

Fig. 16.d Local velocity field at 0.005sec

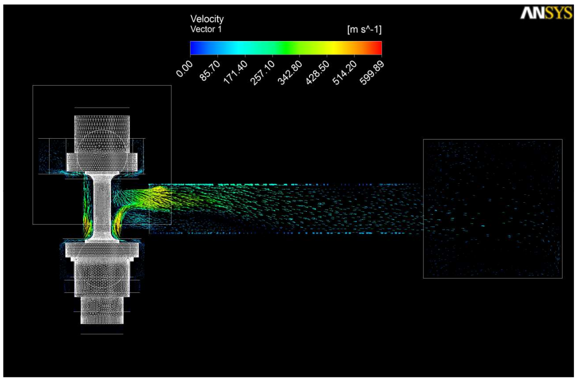

Fig. 16.e Local velocity field at 0.01sec

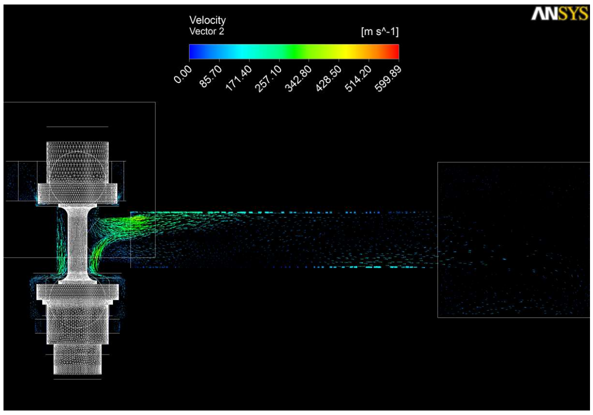

Fig. 16.f Local velocity field at 0.018sec

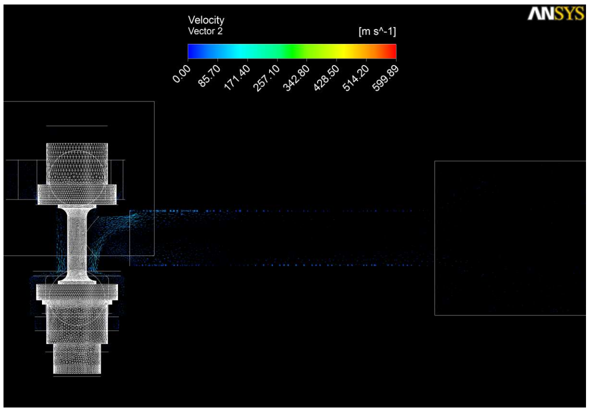

Fig. 16.g Local velocity field at 0.027sec

Compared with pressure field, the velocity vector is almost fully filled in half cycle.

From all the above referred cases we can conclude that the maximum pressure is built up locally and in a short time. Also the high pressure air reaches the cylinder wall in a very short time which satisfies the function of the engine starting valve.

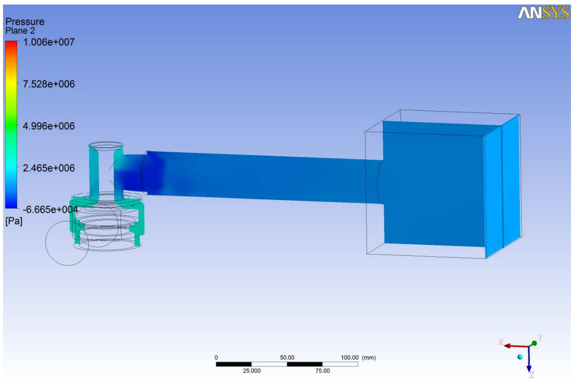

Fig. 17 Pressure at the outlet wall at 0.005 (7.5mm case)

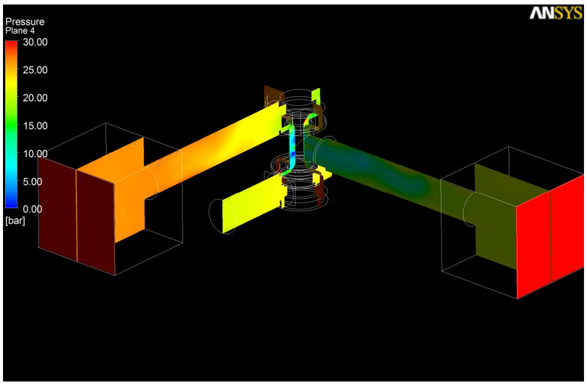

Fig. 18 Pressure at the outlet wall at 0.05s (Sin(t) graph case)

From Fig. 17 and Fig. 18, it can be infer that the high pressure is build up in the outlet wall at about 25 to 30bar at 0.005sec and 0.05sec under the assumption that the high-pressure air pushes the cylinder through the starting valve.

4 Conclusion

In this paper, we have performed numerical simulations to investigate the fluid characteristics of a 3/2 way solenoid valve used in a ship engine. We developed a CFD model using the commercial software ANSYS CFX 12.0. and obtained the preliminary results on the pressure field for the present problem. The three dimensional numerical simulation results will be helpful in understanding the flow characteristics of the valve for the specified industrial application and it can be used for the better design of the valve. The main conclusions drawn from the simulation results are written as follows:

1. Through the present simulation work we could capture the exact flow phenomenon because the modeling is made at 1:1 scale ratio.

2. For an opening pressure of 30bar, it is found that the maximum pressure is very high. But it is accumulated locally in a small area and in very short time. Hence it has marginal effect.

3. In all the cases the pressure stabilization is completed in 0.07sec. Hence we can conclude that the solenoid valve is functioning well.

4. The present study reveals that there is no problem in transporting the high-pressure air to the engine through the solenoid valve which was one of the objectives of the present work.

5. In contrast to other cases, for 30 bar working pressure and spool displacement of 7.5mm, the maximum velocity is low compared to the case with 20 bar as working pressure.

6. In the present work, the more cases is needed like 22,23…,29bar. In the trends of pressure field graph, there was some large gap between the 20bar and 30bar.

7. It will be needed to have the cases by using the many types of codes for the more exact results of the case which has the moving of the spool. And also it will be need to have the experimental results.

Acknowledgements

This work was supported by Technical Center for High-Performance Valves from the Regional Innovation Center (RIC) Program of the Ministry of Knowledge Economy (MKE)

References:

[1]Skousen.P.L, *Valve Hand Book*, McGraw-Hill, 2007.

[2]ANSYS CFX 12.0 User's Manual, ANSYS, Inc.

[3]Menter, F.R., Zonal two equation k-ω turbulence models for aerodynamic flows, *AIAA Paper* 1993, 93-2906.

[4] Kang SM, Lee B-H and Kim T, A numerical study on the flow characteristics through an industrial safety relief valve, *Third IC-EpsMsO*, *Athens*, 8-11 July 2009.