International Journal on Recent and Innovation Trends in Computing and Communication
Volume: 4 Issue: 7
ISSN: 2321-8169
145 - 151

Analysis of Balanced Stiffness Valve by using Transient Finite Element Analysis

Ms. Megharani S. Kashid1
Post Graduate Student
Department of Mechanical Engineering
P.V.P.I.T.Budhgaon, Sangli,India
megha.kashid26@gmail.com

Mr. D. P. Patil2
Associate Professor
Department of Mechanical Engineering
P.V.P.I.T.Budhgaon, Sangli,India
dppatil@pvpitsangli.edu.in

Mr. Vinaay G. Patil3
CEO
Vaftsy CAE
Pune India
vinaay.patil@vasftsycae.com

Abstract— In chemical industries there is necessary to control the flow of liquids between chambers, where it is necessary that valve will be opened when a certain pressure of fluid is reached. To control this fluid flow electronically actuated valves are generally used. Sometimes there is also need of mechanical actuated valve. A single valve will connect three chambers and will control inter flow between these chambers using a balanced stiffness approach where in flow will switch automatically operating at pressure. This paper basically focused on the transient finite element analysis of Balanced Stiffness valve. This transient analysis is generally used to determine the dynamic response of a structure under the action of any general time-dependent loads. It is used to determine the time-varying displacements, stresses, strains, and forces in valve parts as it responds to any transient loads. Here performance of the Balanced Stiffness Valve, i.e. movement of pressure plates observed. Pressure Plate area is exposing to the fluid flow instantaneously as the supply pressure given to pressure plate. Hence it is essential to examine time dependent dynamic response of the valve

Keywords*-Transient finite element analysis; Balanced stiffness valve; transient loads.*

*****

I. INTRODUCTION

Finite Element Analysis is a simulation technique which is used to find out the behavior of components, structures and equipment for various loading conditions such as applied forces, pressures and temperatures. Therefore a complex engineering problem with non-standard shape and geometry can be solved using finite element analysis. The finite element analysis methods result in the stress distribution, displacements and reaction loads at supports etc. for the model..

Balanced stiffness valve is one the most important device to control the flow of two liquids. This valve continuously operated for mixing of two liquids. So, there is necessary to understanding the transient behavior of valve. This paper includes transient finite element analysis of balanced stiffness valve.

II. TRANSIENT FINITE ELEMENT ANALYSIS

A. Input Data:

Inputs received from Vaftsy CAE,Pune. Mentioned in table I and table II.

TABLE I. DATA FOR VALVE DESIGN

Sr.No	Parameter	Intensity
1	Case 1 - Valve operating Pressure from side 1	0.2 Mpa
2	Case 2 - Valve Operating Pressure from side 2	0.185 Mpa
3	Main shell Diameter	100mm
4	Nozzle Diameter	60mm
5	No. of Springs	5

TABLE II. DIMENSIONS OF PARTS

Sr.No.	Part Name and Material		Dimensions
1	Main shell SA 516 Gr 70		Diameter=100mm, Thickness=6mm,Length=135
2	Pressure plate SA 516 Gr 70		Thickness=3mm,Diameter=100 mm
3	Spring ASTM A913-50		Free Length=72 mm,Stiffness=9 N/mm
4	Nozzle SA-106GRB		Diameter=60mm,Thickness=6 mm ,Length=90mm.
5	Reinforcement pad		Outer diameter=80.48
6	Flange SA 516 Gr 70	Main Shell	Inner diameter=100 mm, Outer Diameter=150mm, Thickness = 10mm

ISSN: 2321-8169
145 - 151

		Nozzle	Inner diameter=60 mm, Outer Diameter=90mm, Thickness = 10mm
7	Clit SA 516 Gr 70		Outer diameter = 100 mm ,Inner diameter=95mm Thickness=3mm.
8	Plunger SA 516 Gr 70		Outer Diameter=30 mm, Inner diameter=25mm,Length=36mm , Arm width=15 mm

B .Model

The beginning point of the finite element analysis is the CAD model of the balanced stiffness valve. The modeling is done in ANSYS Workbench 15.Unnecessary details of fillets and chamfers are not included to avoid additional meshing difficulties. Figure 1 shows the CAD model of balanced stiffness valve.

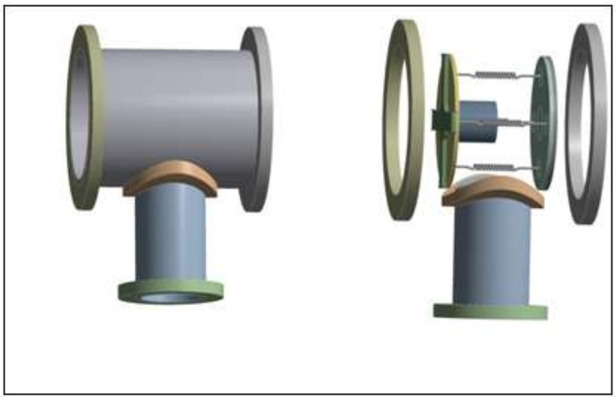


Figure 1.CAD Model of Valve Assembly

C. Contact settings

The contact regions between each valve components are automatically detected by Ansys Workbench 15 when the model assembly is drawn. There are five contact types available in Ansys Workbench 15 for face, node and edge contacts, such as bonded, no separation, frictionless, rough and frictional. The contacts for this finite element analysis of Case 1 and Case 2 are set as follows in Table III and Table IV respectively.

TABLE III. CONTACT TYPES FOR CASE 1

Contact sets	**Contact types**
Flange 1 to shell	Bonded
Shell to Plunger	Bonded
Shell to pressure plate1	Frictionless
Plunger to pressure plate 1	Frictionless
Nozzle to flange	Bonded
Clit to pressure plate2	Bonded

TABLE IV. CONTACT TYPES FOR CASE 2

Contact sets	**Contact types**
Flange 2 to shell	Bonded
Shell to Plunger	Bonded
Shell to pressure plate2	Frictionless
Plunger to pressure plate 1	Bonded
Nozzle to flange	Bonded
Clit to pressure plate2	Frictionless

D. Meshing of Model

SOLID186 is most accurate higher order 20 node hexahedron structural solid element. The element has any spatial orientation. The element includes the anisotropic material properties. The element supports plasticity, hyper elasticity, creep, stress stiffening, large deflection, and large strain capabilities. Meshing details are mentioned in table V.

TABLE V. MESH CONTROL FOR MODEL

Element Type	SOLID 186
Method of mesh control	Hex Dominant
Size	3 mm
Statistics	
No. of nodes	158087
No. of elements	38067

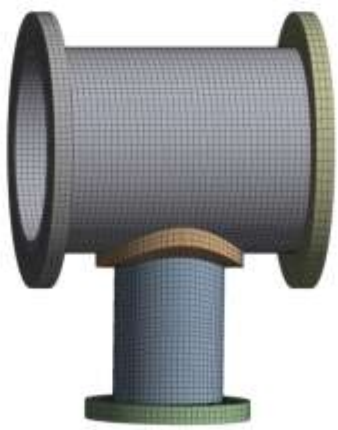


Figure 2. Meshing of model

E. Analysis Procedure

After doing static structural analysis, transient structural analysis is done on the entire valve body. In static analysis load is applied in one step. But for the transient analysis load is applied in 3 steps. Each steps is of 1sec.Plate area exposing to the fluid flow at inlet side is divided into 3 load steps. As pressure at pressure plates side is increases plate moves forward against the spring force and hence at particular time "t" each step is experiencing different pressure. As time advances successive load step is applied on plate. Hence plate is exposed to increasing fluid pressure. Pressure is applied on plate surface as well as inner wall of shell. Table IV indicates the details of transient structural analysis. The sub steps in ANSYS analysis are100 by default. But it has been reduced to 10 in order to reduce the time of analysis.

F. Boundary conditions

Case 1:

The valve is actuated at three pressure level of 0.185 Mpa, 0.185Mpa and 0 Mpa at three time level of 1sec, 2 sec, 3 sec respectively which considering pressure applying in shell and pressure plate 1 . Table VI shows the details of pressure with respect to time.

0 Mpa, 0.185Mpa and 0 Mpa at three times level of 1sec, 2 sec, 3 sec respectively which considering pressure applying in nozzle. Table VII shows the details of pressure with respect to time. Figure 3 shows the boundary conditions for case 1.

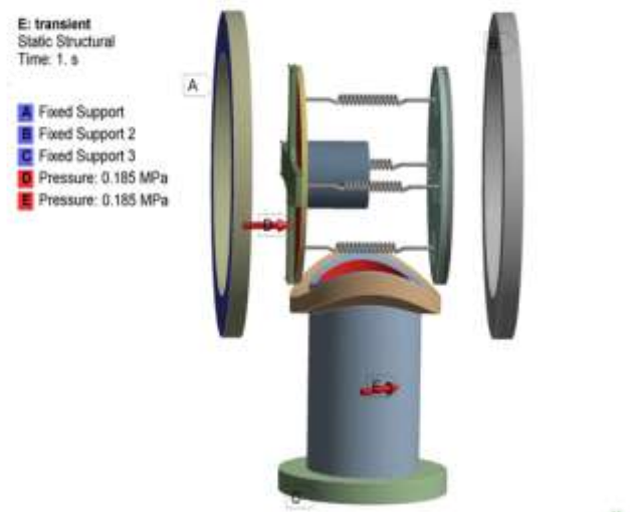


Figure 3. Boundary condition for case 1

TABLE VI . DISTRIBUTION OF PRESSURE ON PRESSURE PLATE 1 WITH RESPECT TO TIME

Steps	Time [s]	Pressure [MPa]
	0	0
1	1	
2	2	0.185
3	3	0

TABLE VII. DISTRIBUTION OF PRESSURE ON NOZZLE WITH RESPECT TO TIME

Steps	Time [s]	Pressure [MPa]
	0	
1	1	0
2	2	0.185
3	3	0

Transient loads are applied on main shell, pressure plate 1 and inner surface of the nozzle. Observed equivalent stresses and deformation for case 1 are shown in below figures.

At 1 sec. pressure acting on pressure plate 1 and inner face of main shell is 0.185 Mpa. Due to this pressure, pressure plate 1 moves forward by compressing 4 springs. In this pressure plate 1 deform with maximum total deformation 36.497mm as shown in the figure 4 by red color. The deformation in the rest of the parts of valve body is very low indicated by blue color.

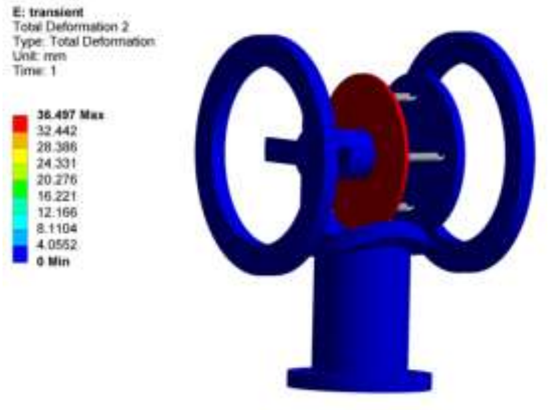


Figure 4. Total Deformation for case 1 at time 1 sec

At 2 sec. pressure acting on pressure plate 1 and inner face of main shell is 0.185 Mpa. In this pressure plate 1 deform with maximum total deformation 36.498 mm as shown in the figure 5 by red color.

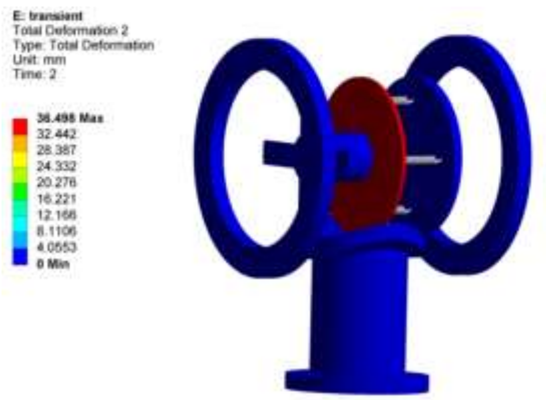


Figure 5.Total Deformation for case 1 at time 2 sec

At 3 sec. pressure acting on pressure plate 1 and inner face of main shell is 0 Mpa, so pressure plate 1 comes to original position due to 4 springs force. In this pressure plate 1 deform with maximum total deformation 0.0080588mm as shown in the figure 6.

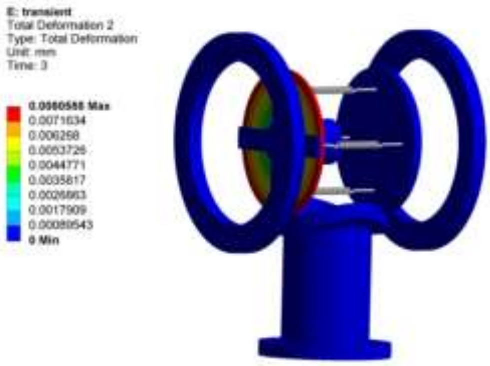


Figure 6. Total deformation for case 1 at time 3sec

At 1 sec. pressure acting on pressure plate 1 and inner face of main shell is 0.185 Mpa. Maximum equivalent stress produced in pressure plate 1 and 2 is 39.263 Mpa.as shown in figure 7 by red color and stresses on the remaining part of the valve body is minimum which is indicated by blue color.

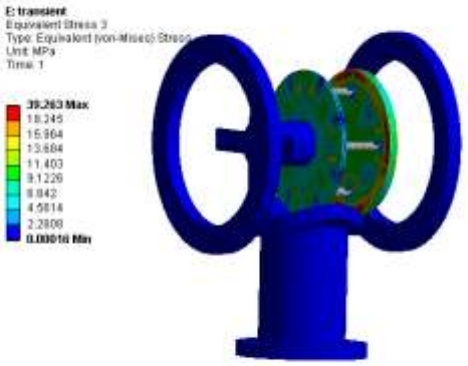


Figure 7. Equivalent (von-Mises) stress for case 1 at 1second

At 2 sec. pressure acting on pressure plate 1 and inner face of main shell is 0.185 Mpa. Maximum equivalent stress produced in pressure plate 1 and 2 is 39.224 Mpa.as shown in figure 8 by red color.

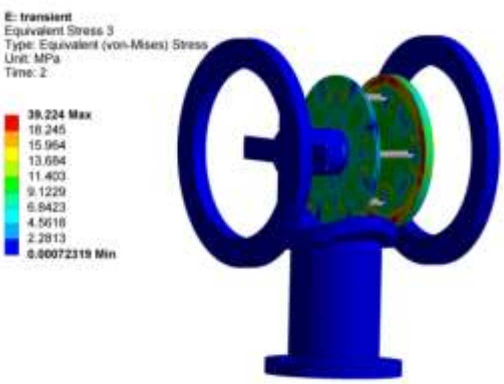


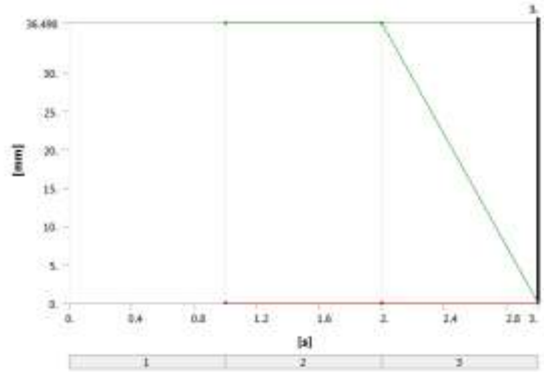
Figure 8. Equivalent (von-Mises) stress for case 1 at 2 sec

At 3 sec. pressure acting on pressure plate 1 and inner face of main shell is 0 Mpa. Maximum equivalent stress produced in pressure plate 1 is 17.168 Mpa.as shown in figure 9 by red color.

Figure 9. Equivalent (von-Mises) stress for case 1 at 3sec

G. Solution Information for case 1

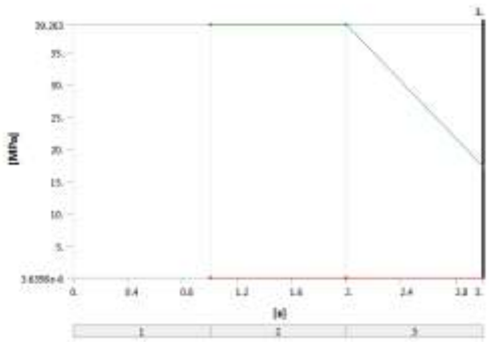
Transient loads are applied on valve in case 1. Equivalent stress and deformation are observed, stresses obtained are within the permissible limit hence the design is safe. Time Dependent Stresses and deformations are tabulated and represented in graph as shown in below.



Graph 1. Total Deformation for case1

TABLE VIII. TOTAL DEFORMATION FOR CASE 1

Time [s]	Minimum [mm]	Maximum [mm]
1		36.497
2		36.498
3	0	8.06E-03



Graph 2. Equivalent stress for case1

TABLE IX. EQUIVALENT STRESS FOR CASE 1

Time [s]	Minimum [MPa]	Maximum [MPa]
1	1.60E-04	39.263
2	7.23E-04	39.224
3	3.64E-06	17.168

Case 2:
The valve is actuated at three pressure level of 0.2 Mpa, 0.2 Mpa and 0 Mpa at three time level of 1sec, 2 sec, 3 sec respectively which considering pressure applying in shell and pressure plate 2 . Table X shows the details of pressure with respect to time.
0 Mpa, 0.2 Mpa and 0 Mpa at three times level of 1sec, 2 sec, 3 sec respectively which considering pressure applying in nozzle. Table XI shows the details of pressure with respect to time. Figure 10 shows the boundary conditions for case 2.

Figure 10.Transient Boundary condition case 2

TABLE X. DISTRIBUTION OF PRESSURE ON PRESSURE PLATE 2 WITH RESPECT TO TIME

Steps	Time [s]	Pressure [MPa]
	0	0
1	1	
2	2	0.2
3	3	0

TABLE XI. DISTRIBUTION OF PRESSURE ON NOZZLE WITH RESPECT TO TIME

Steps	Time [s]	Pressure [MPa]
	0	
1	1	0
2	2	0.2
3	3	0

Transient loads are applied on main shell, pressure plate 2 and inner surface of the nozzle. Observed equivalent stresses and deformation for case 2 are shown in below figures.

At 1 sec. pressure acting on pressure plate 2 and inner face of main shell is 0.185 Mpa. Due to this pressure, pressure plate 2 moves forward by compressing 5 springs. In this pressure plate 2 deform with maximum total deformation 36.782 mm as shown in the figure 11 by red color. The deformation in the rest of the parts of valve body is very low indicated by blue color.

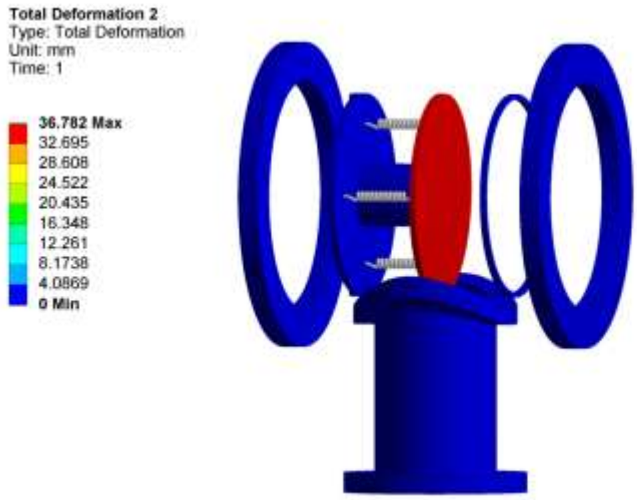


Figure 11. Total deformation for case 2 at 1 sec

At 2 sec. pressure acting on pressure plate 2 and inner face of main shell is 0.2 Mpa. In this pressure plate 2 deform with maximum total deformation 36.786mm as shown in the figure 12 by red color.

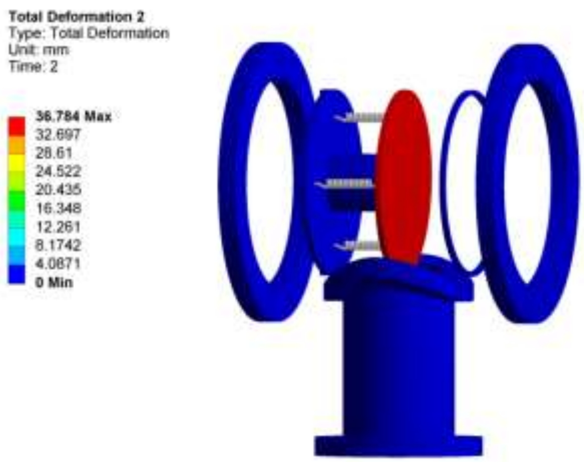


Figure 12. Total deformation for case 2 at 2 sec

At 3 sec. pressure acting on pressure plate 2 and inner face of main shell is 0 Mpa, so pressure plate2 comes to original position due to 5 spring force. In this pressure plate 2 and plunger deform with maximum total deformation 3.0649e-008mm as shown in the figure 13 by red color. The deformation in the rest of the parts of valve body is very low indicated by blue color.

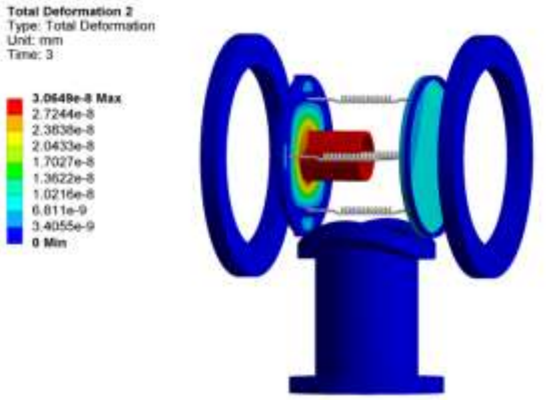


Figure 13. Total deformation for case 2 at 3 sec

At 1 sec. pressure acting on pressure plate 2 and inner face of main shell is 0.2Mpa. Maximum equivalent stress produced in pressure plate 1 and 2 is 38.958 Mpa. as shown in figure 14 by red color and stresses on the remaining part of the valve body is minimum which is indicated by blue color.

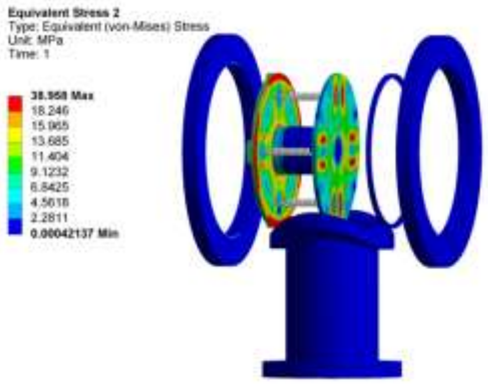


Figure 14. Equivalent (von-Mises) stress for case 2 at 1 sec

At 2 sec. pressure acting on pressure plate 2 and inner face of main shell is 0.2Mpa. Maximum equivalent stress produced in pressure plate 1 and 2 is 38.958 Mpa. as shown in figure 15 by red color and stresses on the remaining part of the valve body is minimum which is indicated by blue color.

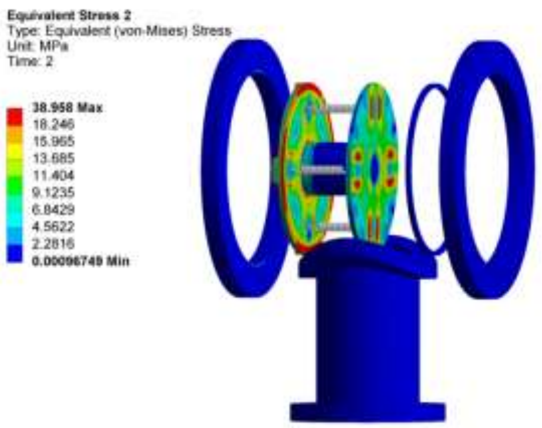


Figure15.Equivalent (von-Mises) stress for case 2 at 2 seconds

At 3 sec. pressure acting on pressure plate 2 and inner face of main shell is 0Mpa. Maximum equivalent stress produced is 0.00025459Mpa.as shown in figure 15.

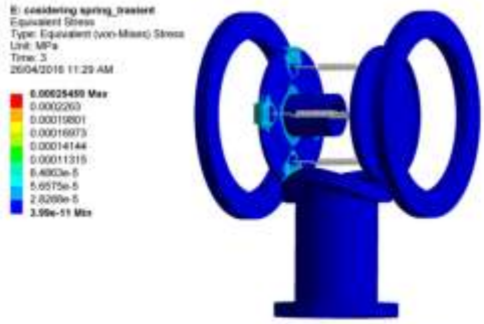
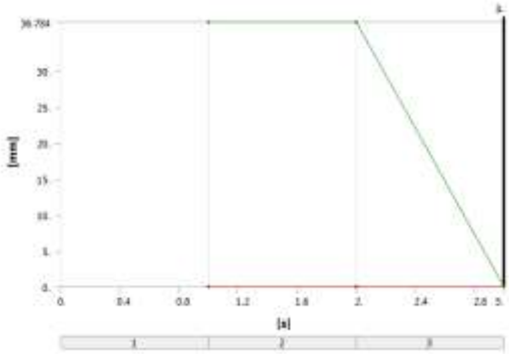


Figure16. Equivalent (von-Mises) stress for case 2 at 3seconds

H. Solution Information for case 2:

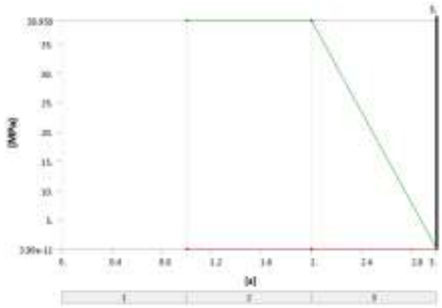
Transient loads are applied on valve in case 2. Equivalent stress and deformation are observed; stresses obtained are within the permissible limit hence the design is safe. Time Dependent Stresses and deformation are tabulated and represented in graph as shown below.



Graph 3. Total Deformation for case2

TABLE XII.TOTAL DEFORMATION FOR CASE 2

Time [s]	Minimum [mm]	Maximum [mm]
1		36.782
2		36.784
3	0	3.06E-08



Graph 4. Equivalent stress of case2

TABLE XII. EQUIVALENT STRESS FOR CASE 2

Time [s]	Minimum [MPa]	Maximum [MPa]
1	4.21E-04	
2	9.67E-04	38.958
3	3.99E-11	2.55E-04

RESULT

TABLE XIII.TRANSIENT STRESSES AND DEFORMATION FOR CASE 1

Sr.No.	Parameter	Time (Sec)	Case 1 Maximum	Case 1 Minimum
1	Total Deformation (mm)	1	36.497	0
		2	36.498	0
		3	8.06E-03	0
2	Equivalent (von-Mises) stress (Mpa)	1	39.263	1.60E-04
		2	39.224	7.23E-04
		3	17.168	3.64E-06

TABLE XIV.TRANSIENT STRESSES AND DEFORMATION FOR CASE 2

Sr.No.	Parameter	Time (Sec)	Case 2 Maximum	Case 2 Minimum
1	Total Deformation (mm)	1	36.782	0
		2	36.784	0
		3	3.06E-08	0
2	Equivalent (von-Mises) stress (Mpa)	1	38.958	4.21E-04
		2	38.958	9.67E-04
		3	2.55E-04	3.99E-11

CONCLUSION

In order to observe performance characteristics of the valve, transient structural analysis is done over valve assembly. This analysis shows maximum stresses developed in case1 & case 2 are within the permissible safety limit also deformation is also sufficient enough for valve performance. From the results we can say valve performs functionally well. Non linear analysis gives more accurate results regarding the stresses

ACKNOWLEDGMENT

I would like to acknowledge Mr. D. P. Patil for helpful discussions related to the work, and Mr. V. G. Patil of Vaftsy Engineering services Ltd. Pune for providing necessary guideline in carrying out Finite Element Analysis.

REFERENCES

[1] Binod Kumar Saha, Himadri Chattopadhyay, Pradipta Basu Mandal, Tapas Gangopadhyay,2014, "Dynamic simulation of a pressure regulating and shut-off valve". Journal of Computers & Fluids 101 (2014) 233–240.

[2] Mr.V.D.Rathod, Prof.G.A.Kadam,2014," Design and Analysis of Pressure Safety Release Valve by using Finite Element Analysis". International Journal of Engineering Trends and Technology (IJETT) – Volume 13 Number 1 – Jul 2014

[3] Sushant M. Patil , Ramchandra G. Desavale , Imran M. Jamadar , 2013,"Conceptual Structure Design Through Thickness Optimization Of High Pressure And High Temperature Self Regulated Pressure Valve Using Non-Linear Transient Finite Element Method". International Journal of Engineering Research & Technology June 2013 Volume 2 Issue 6.

[4] N.N. Manchekar, V. Murali Mohan,2013,"Design of gradual flow reducing valve by finite element analysis". International Journal of Engineering & Science Research June 2013 Volume 3, Issue-6,Pages 257-267.

[5] Ron Darby,2013,"The dynamic response of pressure relief valves in vapor or gas service, part I: Mathematical model". Journal of Loss Prevention in the Process Industries, 2013,1-7.

[6] Xue-Guan Song, Young-Chul Park, Joon-Hong Park,2013, "Blowdown prediction of a conventional pressure relief valve with a simplified dynamic model". Mathematical and Computer Modelling 57 (2013) 279–288.

[7] A.Beune,J.G.M.Kuerten,M.P.C. van Heumen,2012 "CFD analysis with fluid–structure interaction of opening high–pressure safety valves."Computers & Fluids 64 (2012) 108-116.

[8] A.J.Ortega, B. N. Azevedo, L. F. G. Pires, A. O. Nieckele, L. F. A. Azevedo,2009, "Analysis of the discharge coefficient of a spring loaded pressure relief valve during its dynamic behavior". Journal of International Congress of Mechanical Engineering November 15-20, 2009.