Design and Analysis of Balanced Stiffness Valve by using Finite Element Analysis

[1]Ms.Megharani.S.Kashid ,[2]Mr..D.P.Patil,[3]Mr.A.A.Kumbhojkar

[1]P.G.Student,[2]Associate Professor,[3]Associate Professor

[1]Mechanical Engineering Department, PadmabhooshanVasantdadaPatil Institute of Technology,Budhgaon,India

---

***Abstract* -Modern industries like chemical, involves so many processes require regulating the flow of liquid and usually the parameter of concern is the pressure. In these applications, it is very desirable to have a means to balance the pressure of two different feeds of fluid which are to be mixed. There are some reaction in which we cannot mix two chemicals at a time because if we mix at a time the explosion occurs. So there is need to supply chemicals one by one. There are many valves available in market to control the supply of fluid, however the intent of this design project is have a total mechanical system, which has an in built response mechanism. More than flow regulation the requirement is to design a valve which automatically switches from one opening to another based on the pressure levels. The idea behind this project is to introduce regulation of flow of liquid. This will require a complex construction and immaculate calculation of how the stiffness's behave and how they need to be balanced to create a precise timing. In this paper finite element analysis has been introduced in order to finalize the geometrical parameter of Balanced Stiffness valve. However key constraints in designing the valve are geometrical parameters as well as operating parameters. These parameters are analyzed by FEA.**

***Index Terms* - Balanced Stiffness Valve, FEA.**

I. INTRODUCTION

Many electronic, pneumatic and hydraulic systems exist today to control fluid based on pressure. Each of system requires a power source of some type, such as electricity or compressed air in order to operate. There are considerably insufficient works done in the field of flow control valve which is capable of operating at all times, especially during a period of power failure and when system controls are nonfunctional. So in many industrial applications, where it is very desirable to have a means to balance the pressure of two different feeds of fluid which are to be mixed, for this there is a need of mechanically operated balanced stiffness valve.

- ***Problem Statement***

In the current system there are two chambers containing fluids. The flow of fluids from these chambers to the third chamber is regulated using two separate electronically operated valves. It is proposed to substitute these two electronically operated valves by a mechanically operated balanced stiffness valve. This mechanically operated balanced stiffness valve is to be designed as shown in fig.1 which will connect the two chambers containing different fluids to a common chamber-the third chamber mentioned above where the fluids will be mixed. This project focuses on the flow of liquid between chambers, where it is required that flow be shut off when a certain pressure is reached. 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. The idea is to make the operation more simple, reliable and economically.

II. VALVE DESIGN

To go through the overall design components of Balanced Stiffness Valve need refer ASME codes, Ansys 15 software, mathematic equations etc.

A. Input data

Inlet loop has been finalized as per the detail dimensions given in following Table1and dimensions has been finalized as per the ASME code. The details of dimensions are given in following Table1.

Table 1: Given Data for Valve Design

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

B .Design of main shell

Material for Main shell: SA 516 Gr 70

According to ASME Section-VIII, Division-I, UG27, Required Thickness due to Internal Pressure

$$\mathbf{t_r} = \frac{\mathbf{P.R}}{\mathbf{(S.E-0.6P)}} + \textbf{Corrosion Allowance} \quad (1)$$

Where
tr = Required Thickness for shell
P = Max operating Pressure
S = Allowable Stress
E = Modulus of Elasticity

$$t_r = \frac{0.200 \text{ x } 50}{[(138.023 \text{ x } 1)-(0.6 \text{ x } 0.200)]} + 3$$

t_r = 0.0725+3
tr = 3.0725 mm
3.1 mm plate is not available. Hence if thickness exceeds 3mm then next available is 6mm. So the thickness is modified as 6mm.
Length of main shell = L =1.35 X Internal Diameter of shell
= 1.35 X 100
= 135 mm.

C. Design of pressure plate

Material for Pressure Plate: SA 516 Gr 70

For design of plate, ASME code is used to determine the thickness of the plate
Referring ASME section VIII, div-I,

$$\mathbf{t= d}\sqrt{\frac{C.P}{S.E}} \quad (2)$$

Where,
t = thickness of circular plate
C = factor considering the method of attachment
d = diameter of the vessel.
p = internal pressure
S = Allowable stress

$$t = \mathbf{100 \text{ x }} \sqrt{\frac{\mathbf{0.2X0.2}}{\mathbf{138.023X1}}}$$

t = 100 x 0.0170237
t = 1.70237 mm

D. Spring design

Material for spring: ASTM A913-50
Deflection of spring is (δ),
(δ) = 36mm
Analytical design of spring does not consider friction.
Determine the force acting on spring
P = 0.2Mpa
F = Pressure ×Area (3)
$= 0.2 \text{ X } \frac{\pi}{4} X ds^2$
=1570 N
Force acting on plate = F = 1570.796 N
Force acting on each spring = f = $\frac{1570.796.}{5}$
= 314.16 N
We know,
Stiffness k = F/δ (4)
$k = \frac{314.16.}{36}$
k = 8.727 N/mm.
Stiffness of a spring=8.727 N/mm

E. Nozzle design

For design of Nozzle, ASME code is used to determine the thickness of the Nozzle
Referring ASME section VIII, div-I,
Material for nozzle: SA-106 GRB

$$tr = \frac{P.R}{(S.E - 0.6P)} + \text{Corrosion Allowance} \quad (5)$$

Where
tr= Required Thickness
P= Max operating Pressure
S= Allowable Stress
E= Modulus Of Elasticity

$$tr = \frac{0.2 \times 30}{(118 \times 1 - 0.6 \times 0.200)} + 3$$

tr= 0.0509+3
tr= 3.0509 mm
3.0509 mm thickness is not available. Hence if thickness exceeds 3mm then next available is 6mm. So the thickness is modified as 6mm.

Parameters required for Area calculations:

Nozzle diameter
D_n= 60mm
R_n= 30 mm

Nozzle Corrosion allowance
η_{ca}=3mm

$$t_n = N_{wall} - nca \quad (6)$$
$$= 6 - 3$$
$$t_n = 3 \text{ mm}$$

According to ASME code Section VIII, UG45, Minimum thickness excluding corrosion allowance is 1.5 mm

$$T_{reqN} = 1.5 + nca \text{ mm} \quad (7)$$
$$T_{reqN} = 1.5 + 3$$
$$T_{reqN} = 4.5 \text{ mm}$$

Among both values of T_{reqN} , $T_{reqN} = 4.5$ mm is maximum value.
Hence,
$T_{reqN} = 4.5$ mm
Exterior projection of nozzle

$$L_p = 1.5 \times dn \quad (8)$$
$$= 1.5 \times 60$$
$$L_p = 90 \text{ mm}$$

Interior projection of nozzle lp = 0 mm

$$t_{rnR} = T_{reqN} - nca \quad (9)$$
$$= 4.5 - 3$$
$$= 1.5 \text{ mm}$$

$F_{r1} = F_{r2} = 1$
Since, we are using same material for shell and wall.

$$ti = N_{wall} - 2 \times nca \quad (10)$$
$$= 6 - 2 \times 3$$
$$t_i = 0$$

Area calculations for nozzle:

$$A_r = 1 \times d \times T_{req} + 2 \times t_n \times T_{req} \times (1 - F_{r1}) \quad (11)$$
$$= 1 \times 60 \times 3.0725 + 2 \times 3 \times 3.0725 \times (1 - 1)$$
$$A_r = 184.35 \text{ mm}^2$$

$$A_1 = \max(d, 2 \times (n_t + t_n)) \times (E_1 \times n_t - 1 \times T_{req}) - 2t_n \times (E_1 \times n_t - T_{req}) \times (1 - F_{r1}) \quad (12)$$
$$= \max(60, 2 \times (6 + 3)) \times (1 \times 6 - 1 \times 3.0725) - 2 \times 3 \times (1 \times 6 - 3.0725) \times (1 - 1)$$
$$= 60 \times (1 \times 6 - 1 \times 3.0725) - 6 \times (6 - 3.0725) \times (1 - 1)$$
$$A_1 = 158.085 \text{ mm}^2$$

$$A_2 = \min[(t_n - t_{rnR}) \times F_{r2} \times \min(5n_t, 2 \times L_p), (t_n - t_{rnR}) \times F_{r2} \times \min(5 \times t_n, 2 \times L_p)] \quad (13)$$
$$= \min[(3 - 1.5) \times 1 \times \min(5 \times 6, 2 \times 90), (3 - 1.5) \times 1 \times \min(5 \times 3, 2 \times 90)]$$
$$= \min(45, 22.5)$$
$$A_2 = 22.5 \text{ mm}^2$$

$A_3 = \min[5 \times n_t \times t_i \times F_{r2}, 5 \times t_i^2 \times F_{r2}, 2 \times h \times t_i \times F_{r2}]$ (14)
$A_3 = \min[5 \times 3 \times 0 \times 1, 5 \times 0 \times 1, 2 \times 0 \times 0 \times 1]$
$= \min(0,0,0)$
$A_3 = 0$
$A_a = A_1 + A_2 + A_3 = 158.085 + 22.5 + 0$ (15)
$A_a = 180.585 \text{ mm}^2$
Here, $A_a < A_r$, Hence reinforcing pad is required. The nozzle is not self-reinforced nozzle.
Reinforcing pad design
Volume of Hole = Volume of reinforcement Pad
$\pi r^2 X \text{ ts} = (\pi Ro^2 - \pi R^2) X t_p$ (16)
$\pi (30)^2 X 6 = (Ro^2 - (30)^2) X 1.25X6$
Ro = 40.24mm.

F. Flange design

a. Main shell flange

- Design Pressure= 0.2MPa
- Flange outer Diameter=1.5Xds=1.5X100=150mm (17)
- Flange outer Diameter= ds =100
- Thickness of flange =10mm.

b. Nozzle flange

- Design Pressure= 0.2MPa
- Flange outer Diameter=1.5Xd_N=1.5X60=90mm (18)
- Flange inner Diameter=d_N=60
- Thickness of flange =t=10 mm

G. Clit design:

1. Clit outer Diameter=100mm.
2. Clit inner Diameter=95mm.
3. Clit thickness=t_1

t_1 = 3 % X Plate diameter (19)
$= \frac{3 X 100}{100}$.
= 3 mm.

H. Plunger design:

- Diameter = 0.25 X plate diameter (20)
  = 0.25 X 100
  = 25mm.
- Length = 36 mm
- Arm width = 15 % X plate diameter (21)
  = 0.15 X 100
  = 15mm.

For fitting the valve, different components of the valve are designed for dimension. The names and dimensions of the component are given in Table 2.

Table 2: Detail Dimensions of parts of valve

Sr.No.	Part Name	Dimensions
1	Main shell	Diameter=100mm,Thickness=6mm,Length=135
2	Pressure plate	Thickness=3 mm, Diameter=100mm
3	Spring	Free Length=72 mm, Stiffness=9 N/mm
4	Nozzle	Diameter=60mm,Thickness=6 mm,Length=90mm.
5	Reinforcement pad	Outer diameter=80.48

6	Flange	Main Shell	Inner diameter=100 mm, Outer Diameter=150mm,Thickness = 10mm
		Nozzle	Inner diameter=60 mm, Outer Diameter=90mm,Thickness = 10mm
7	Clit		Outer diameter = 100 mm, Inner diameter=95mm Thickness=3mm.
8	Plunger		Diameter=25 mm, Length=36mm,Arm width=15 mm

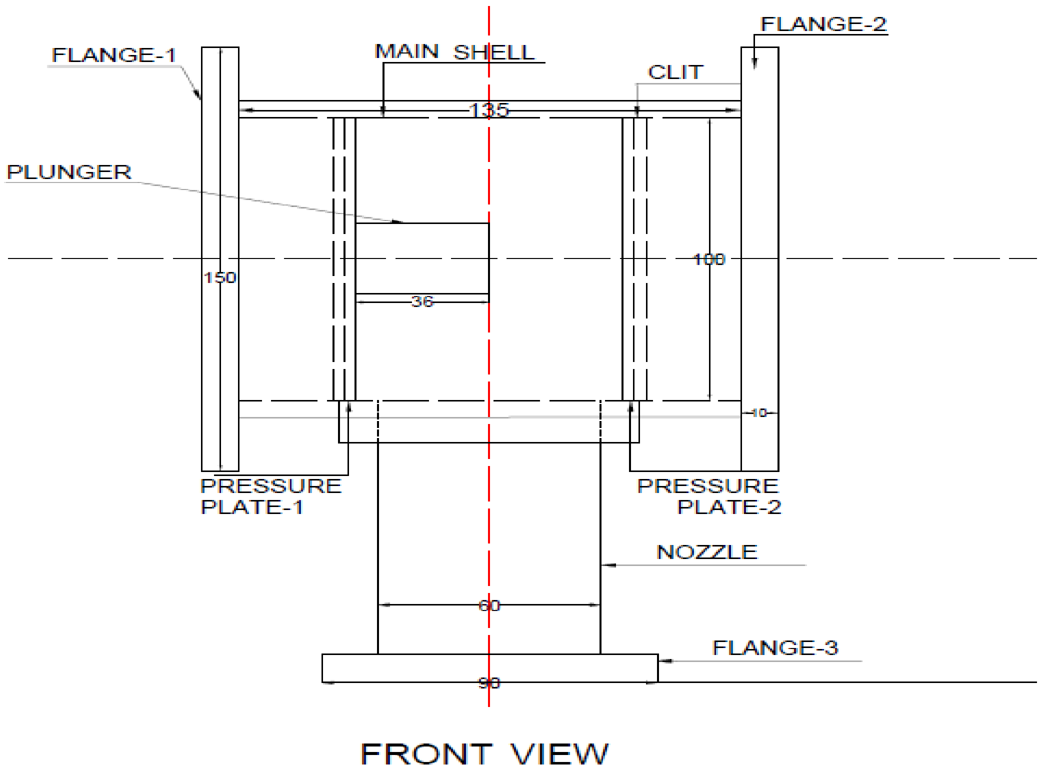

Figure1: Scaled Drawing for Balanced stiffness Valve Model

IV. ANALYSIS TYPE- STATIC STRUCTURAL ANALYSIS

To check the safety of entire assembly it is important to carry out steady state structural analysis. All the valve components designed in previous sections are modeled in ANSYS workbench. Also meshing of the assembly model is done with SOLID 186 element. Further details of analysis are given below.

Table 3: Cases for FEA analysis

Sr.No.	Case	Condition
1	Case 1	Flange 1, Flange 2, Flange 3 are fixed and pressure 0.185 MPa applied on pressure plate 1.
2	Case 2	Flange 1, Flange 2, Flange 3 are fixed and pressure 0.2 MPa applied on pressure plate 2.

A. 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.

Model consists of following parts:-

a) Main shell e) Nozzle
b) Pressure plate f) Reinforcement pad
c) Spring g) Plunger
d) Flange h) Clit

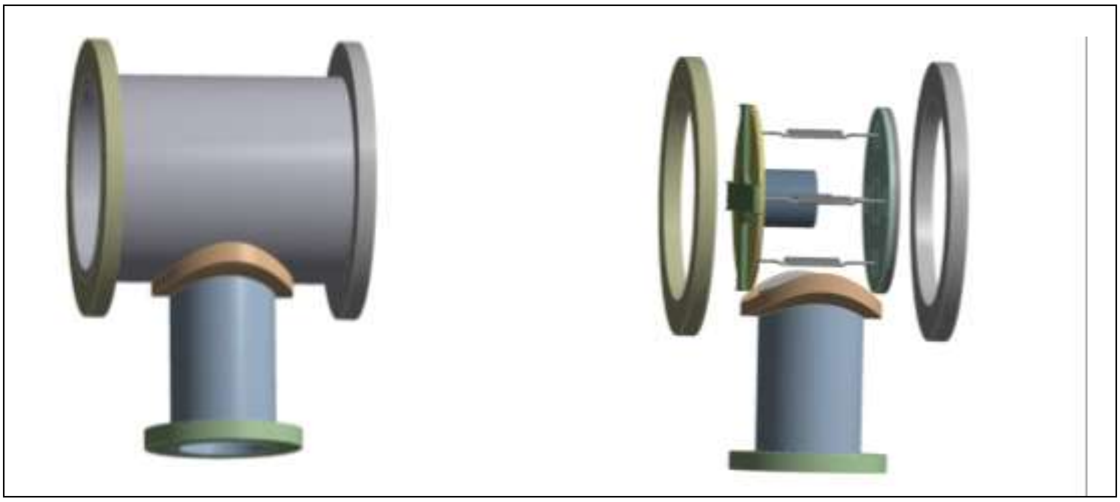

Figure2: CAD Model of Valve Assembly

B. 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 contact settings for this finite element analysis Case 1 and Case 2 are set as follows in Table 4and Table 5 respectively.

Table 4: 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

Table5: 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

C. Meshing of model

SOLID186 is a higher order 20 node hexahedron structural solid element is used for meshing. The element has any spatial orientation. The element input data includes the anisotropic material properties. The element supports plasticity, hyper elasticity, creep, stress stiffening, large deflection, and large strain capabilities. Size of elements influences the convergence of the solution directly and hence it has to be chosen with care. If the size of elements is small, the final solution is expected to be more accurate. However, it should be remembered that the use of elements of smaller size will also mean more computational time. As the number of elements increases, the size of each element must decrease, and consequently the accuracy of the model generally increases.

Table 6: 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

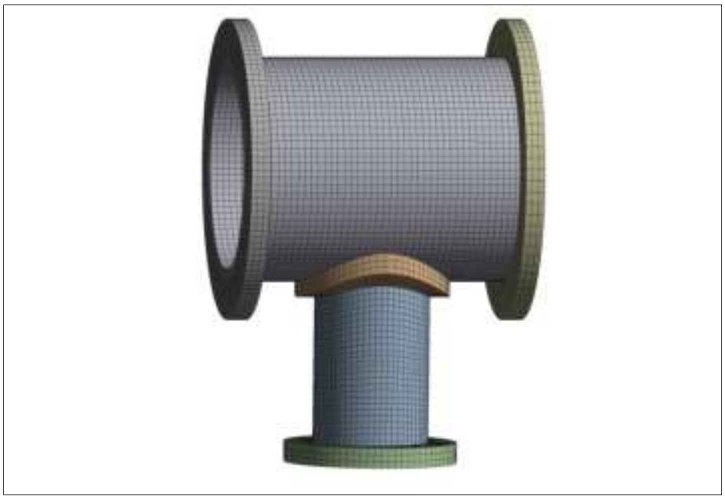

Figure3: Meshing of model

D. Boundary conditions of balanced stiffness valve assembly

For structural analysis of balanced stiffness valve it is required to apply boundary conditions on model. There two cases are considered for applying boundary conditions as mentioned in table 7.Also these boundary conditions are shown in figure 4 and figure5 for case 1 and case 2 respectively.

Table 7: Boundary conditions of balanced stiffness valve assembly

Sr.No.	Case	Boundary Conditions
1	Case 1	Flange 1, Flange 2, Flange 3 are fixed and pressure 0.185 MPa applied on pressure plate 1.
2	Case 2	Flange 1, Flange 2, Flange 3 are fixed and pressure 0.2 MPa applied on pressure plate 2.

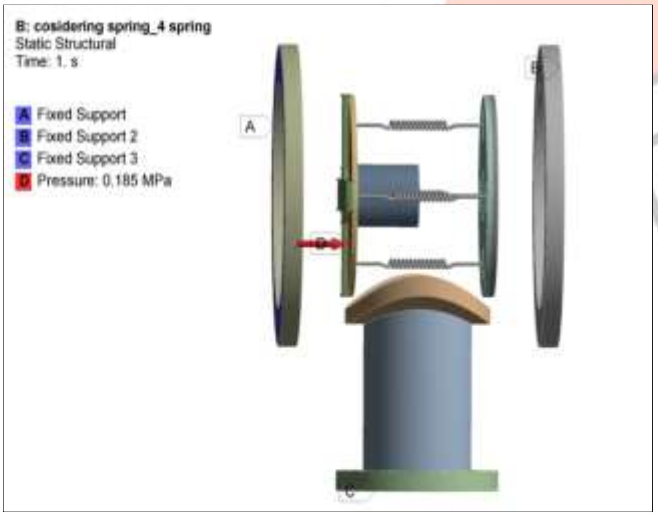

Figure 4: Static Analysis Boundary condition for case 1

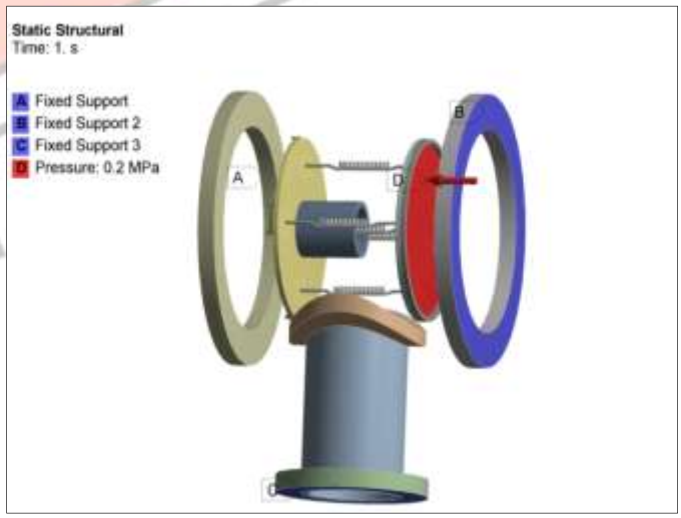

Figure 5: Static Analysis Boundary condition for case 2

E. Solution information:

After giving the boundary conditions the model is solved for

- Dimensions of valve body parts
- Equivalent stress (Von-misses stress)
- Total deformation

a. Check for dimension

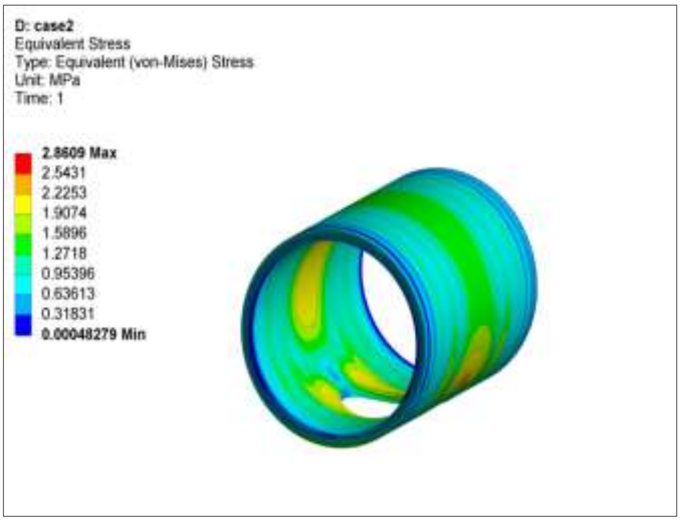

Figure 6: Equivalent stresses in main shell

Figure 6 shows equivalent von- Mises stress in main shell its maximum value is 2.8609 Mpa. The stresses in pipe are within the permissible limit. So the 6 mm calculated thickness of main shell which is safe from dimensional point of view.

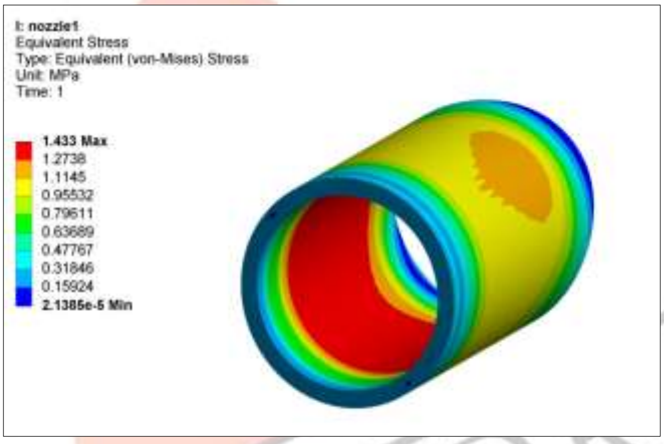

Figure 7: Equivalent stresses in Nozzle

Figure7 shows equivalent von- Mises stress in nozzle its maximum value is 1.433MPa .The stresses in nozzle are within the permissible limit .So the calculated 6mm thickness of nozzle is safe from dimensional point of view.

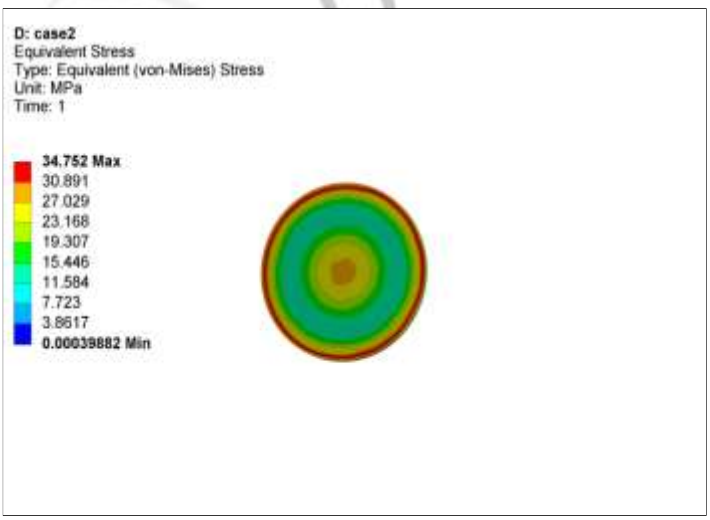

Figure 8: Equivalent stresses in pressure plate

Figure8 shows equivalent von - Mises stress in pressure plate its maximum value is 34.752MPa. The stresses in pressure plate are within the permissible limit. So calculated 3mm thickness of the pressure plate is safe from dimensional point of view

Table 8: Analytical Vs Theoretical Stresses

Sr No	Component	Analytical Stress(Mpa)	Design Stress(Mpa)
1	Main Shell	2.8609	137.89515
2	Nozzle	1.433	117.90035
3	Pressure Plate	34.752	137.89515

Table 8 shows the equivalent stresses produced in main shell, nozzle and pressure plate after finite element analysis done for dimensional check. From above table it is been observed that analytical stresses are less than the design stresses .So the design is safe from the dimensional point of view.

b. Full model analysis

After dimensions check of parts of valve further analysis has been done for the full geometry of the valve. FCorresponding equivalent stresses and deformation are observed in case 1 and case 2 which are 38.626MPa and 38.958 respectively. These stresses are within the permissible limits also deformation is also sufficient enough for valve performance. Hence design is safe.

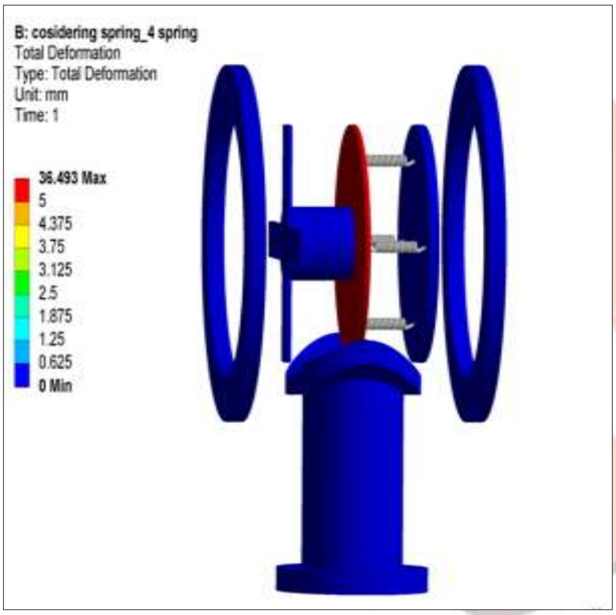

Figure 9: Total Deformation for case 1

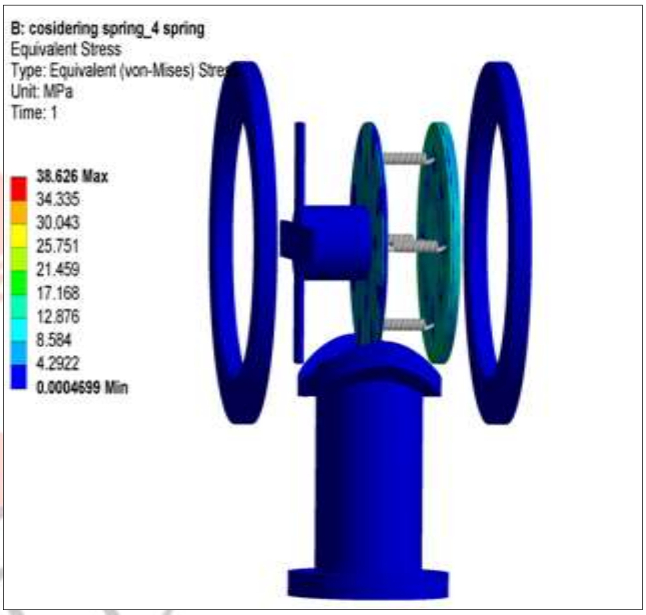

Figure 10: Equivalent (von-Mises) stress for case 1

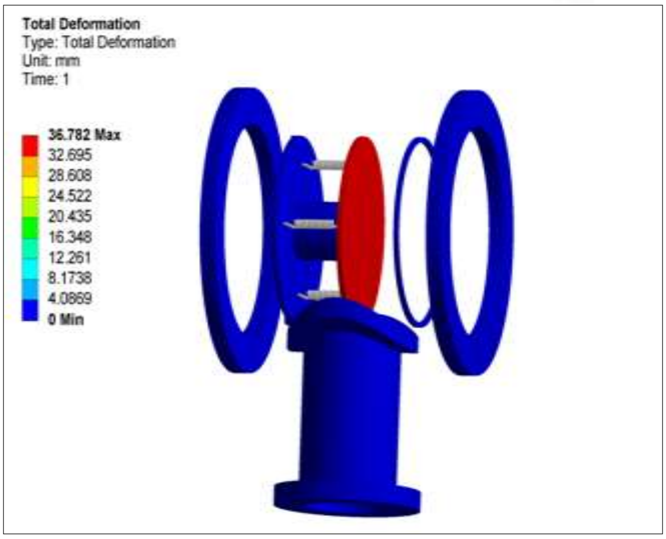

Figure 11 : Total Deformation for case 2

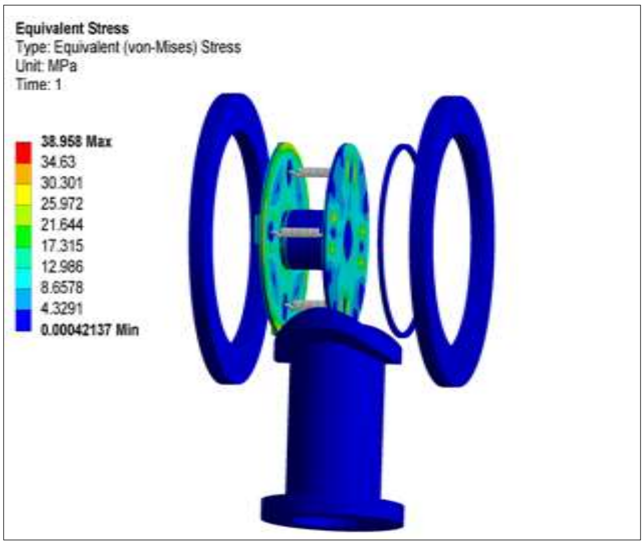

Figure 12 : Equivalent(von-Mises) stress for case 2

F. Result of Finite Element Analysis:

Table 9: Result of Finite Element Analysis

Sr.No.	Parameter	Case 1		Case 2	
		Max	Min	Max	Min
1	Total Deformation (mm)	36.493	0	36.782	0
2	Equivalent (von-Mises) stress (Mpa)	38.626	0.0004699	38.958	0.00042137

Table 9 shows the result of finite element analysis. In this maximum and minimum total deformation and equivalent stresses for case 1 and case 2 are mentioned.

G. DISCUSSIONS ON RESULTS AND CONCLUSIONS

1. The outputs obtained from structural analysis are equivalent stress and total deformation. From structural point of view it is necessary that the stress induced in valve must be less than design stress.
2. From analysis it is been observed that for case 1 and case 2 maximum equivalent stresses are 38.626 Mpa and 38.958 Mpa respectively which are less than design stress. So the design is safe.
3. Total deformations observed for case 1 and case 2 are 36.493 and 36.782 mm. respectively which is sufficient for the performance of the balanced stiffness Valve.

I.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.
[9] “CASTI Guidebook Series - Volume 4 to ASME Section VIII Division 1 for Pressure Vessels” Bruce E. Ball, Will J. Carter, Third Edition (Covering 2001 Edition ofASME Section VIII Division 1).
[10]Introduction to“CHEMICAL EQUIPMENT DESIGN” Mechanical AspectsB.C.Bhattacharyya Indian Institute of Technology Kharagpur CBS Publishers & Distributors Pvt.Ltd.
[11] “Advance Mechanics of Materials” Arthur P. Boresi, Richard J. Schmidt, John Wiley & sons, Inc; sixth edition.
[12] “Pressure Vessel Design” Guides and Procedures, P.V. Engineering.