IJRAR.ORG E-ISSN: 2348-1269, P-ISSN: 2349-5138

INTERNATIONAL JOURNAL OF RESEARCH AND ANALYTICAL REVIEWS (IJRAR) | IJRAR.ORG

An International Open Access, Peer-reviewed, Refereed Journal

FINITE ELEMENT ANALYSIS OF 10MT VALVE TESTING EQUIPMENT

[1]Name of 1st Author : Siddappa Y Babaleshwar, [2]Name of 2nd Author : R.H. Angadi,

[3]Name of 3rd Author : Shivaling I Mukanavar

[1]Designation of 1st Author : Senior scale lecturer, [2]Designation of 2nd Author : Assistant Professor, [3]Designation of 3rd Author : Selection grade I lecturer

[1]Name of Department of 1st Author : Mechanical Engineering

[1]Name of organization of 1st Author : Government Polytechnic Rabakavi-Banahatti, Bagalkote, India

Abstract : M/s. MSM HYDROPNEUMATIC ENGINEERS Belgaum is involved in the manufacture of different range of SPM's. The company's main aim is to reduce the cost of the cost of production without compromising on the quality of the output. Using the optimum resources possible in designing the SPM's an attempt has been made in this direction to reduce the cost of production of valve testing equipment.

So here we consider an industrial application project consisting o f "**Finite Element Analysis of 10T Valve Testing Equipment"** This has to compensate the forces acting on the Component and has to fulfill certain critical constraints.

Solid works is the software has been used for creating 2D and 3D of assembly and parts and Analysis.

I. INTRODUCTION

Designing the hydraulic system based on principles of hydraulics such as in aero plane landing gears, hydraulic lifts, press equipments, test equipments hydraulic jacks Understanding hydraulic principles is essential to know how these systems function. Hydraulics is used for lifting heavy loads and in equipment for construction, mining agriculture, and material handing

1.2 USES OF HYDRAULIC SYSTEMS

The hydraulic systems are divided into two sciences:

- Hydrostatics
- Hydrodynamics

Hydrostatics

Hydrostatics involves forcing fluid within a closed container to transmit power. For example, when using a hydraulic jack to lift a car, the liquid is displaced so the jack extends upward. The following are applications:

- Hydraulic cylinder actuation.
- Hydraulic jack, press.

Hydrodynamics

Following are examples for hydrodynamics

- Torque converter
- In turbine water is used to rotate wheel, pumps.

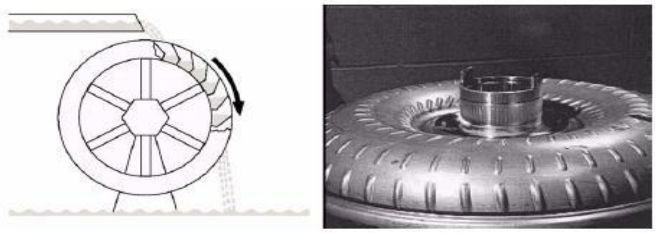


Figure 1: Hydrodynamics

1.3 PRINCIPLES OF HYDRAULIC

The various benefits of using fluid

1. Liquids are practically incompressible.
2. Liquids apply in all direction equal pressure.
3. Liquids takes shape of the object in which it is stored.

Practically liquids are non compressible

Hydraulic oil is ideal for hydraulic machine because of its non compressible nature. Compressing a object reduces the space it occupies. airis not suitable for hydraulic systems as air compress and under pressure takes less space. A liquid maintains the same volume when external force is applied [7].

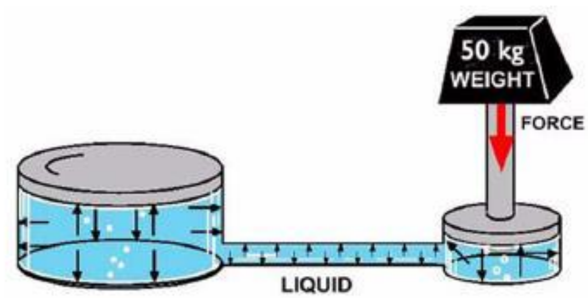
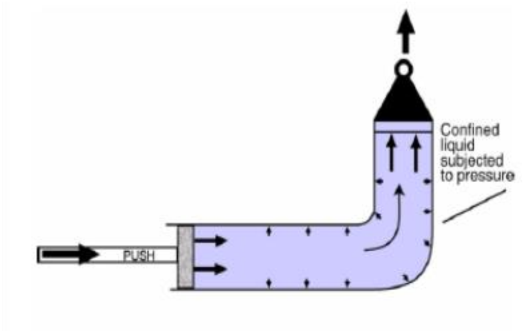


Figure 2: Practically liquids are non compressible

Pressure apply in all direction equally by fluid



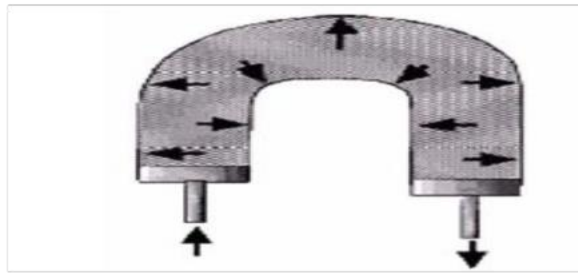


Figure: 3 Figure: 4

Figure 3&4: Pressure apply in all direction equally by fluid

The pressure reading in hydraulic system line is same no matter where pressure gauge is fixed, as shown in fig 4. When pressure is applied at one end the amount of pressure exerted equally on the entire pipe line same displacement one end to the other end as per the Pascal's law same principle is applied in hydraulic jack where less input force output is more.

Liquids take shape of container

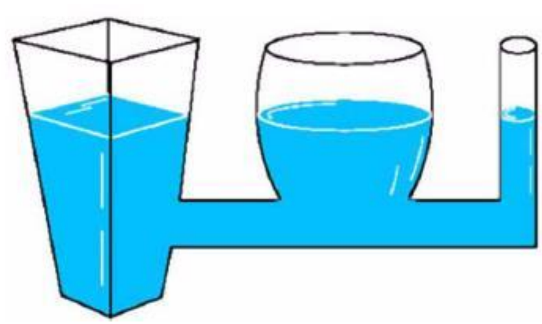


Figure 5: Liquids take shape of container

Shown in Figure 5 all three containers are differing in their geometry but the level of the liquid found is same.

4

1.4 FLUID POWER

According to Pascal's law, when force is applied at the top of the closed container which is filled with liquid the same pressure is exerted on the all boundary of the container as shown in the

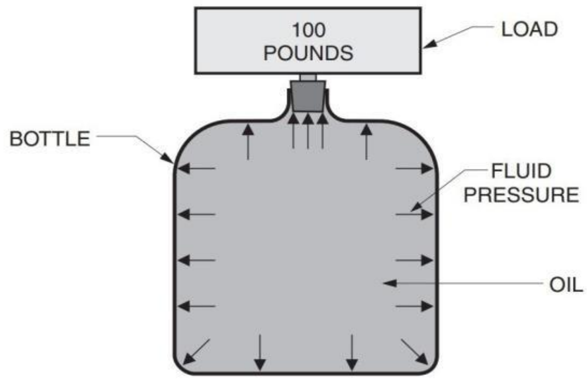


Figure 6

Figure 6: Force applied to the liquid

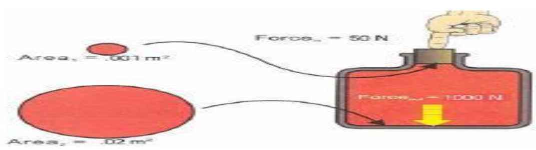


Figure7: Container bursting due to pressure.

Since practically liquids found as incompressible, forces applied to them are transmitted without reduction through the entire liquid and exert equal pressure on all areas equally on the container. Because the bottle's body has a much larger surface area than neck, even a small force on the stopper can cause the body to rupture. The phenomenon is shown inFigure7

Figure 8: Pressure, area, force relationship

Figure 8 shows the relation between the force, area and pressure Force = Pressure$\times$ Area When force of 50N applied on small area 0.001 m^2pressure exerted is calculated by using above relation

Pressure = Force /Area = 50/0.001= 50000 N/m^2 when same pressure applied on big area 0.02m^2 force exerted is calculated as F= P × A = 50000x 0.02 = 1000N

1.5HYDRAULIC CIRCUIT

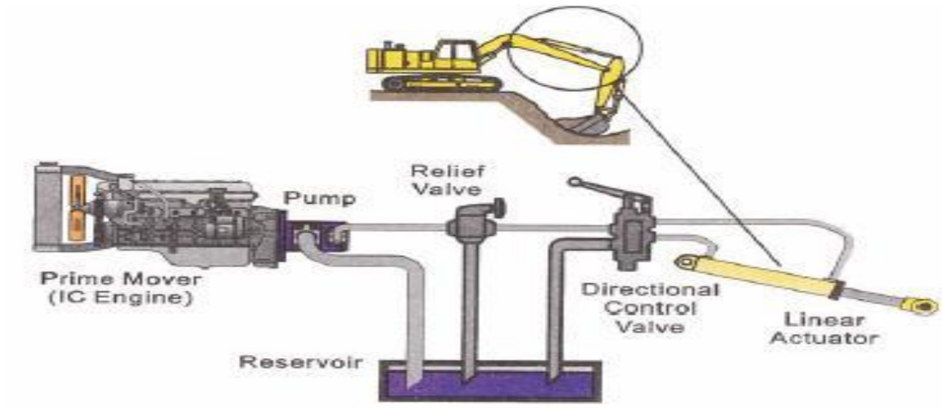


Figure9: Hydraulic Circuit

Figure 9 shows the simple hydraulic circuit which includes power pack which includes reservoir Prime mover electrical motor or I C engine which is coupled to the hydraulic pumpnext in line relief valve is connected to regulate the working pressure the action of linear actuator is controlled by direction control valve it maybe hand operated or electrical as per the customers requirement

1.6 PRESSURE

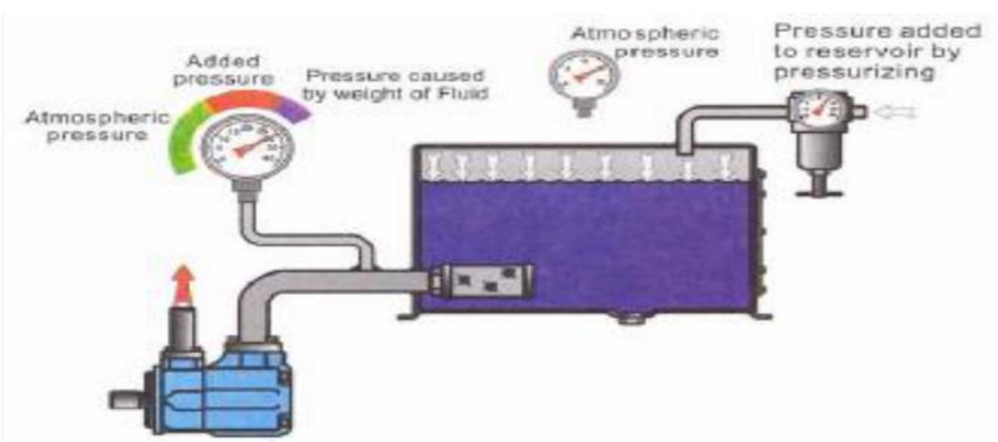


Figure 10: Pressure at exit of power pack

As shown in Figure 10 the pressure at the exit of the power pack is regulated using the pressure regulator as per the device working pressure

1.7BAROMETRIC PRESSURE

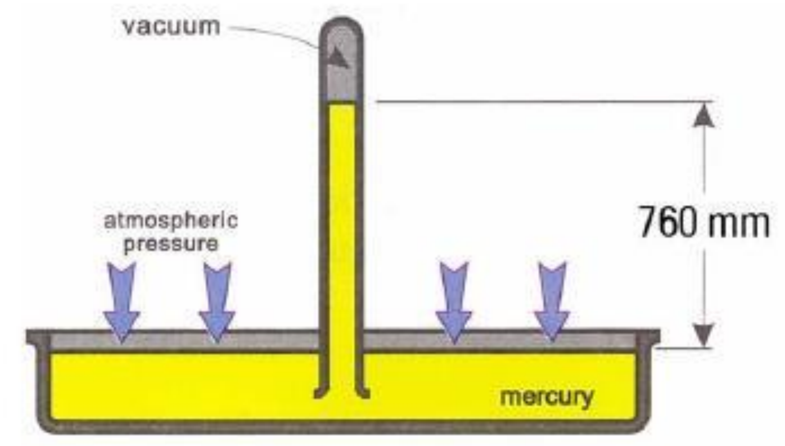


Figure11: Barometer principle

Figure 11 shows how the atmospheric pressure is experimentally measured by using glass tube and mercury when long glass tube is inserted in to the mercury tank the mercury raise up to 760mm and stop that is atmospheric pressure is 760mmHg.

1.8FLOW (Q)

When external force or pressure is exerted on liquid or gases it tend to move from one end to the other end this is known as flow Q=AxV quantity of flow is depends on area of the pipe and the velocity

1.8.1 LAMINAR FLOW

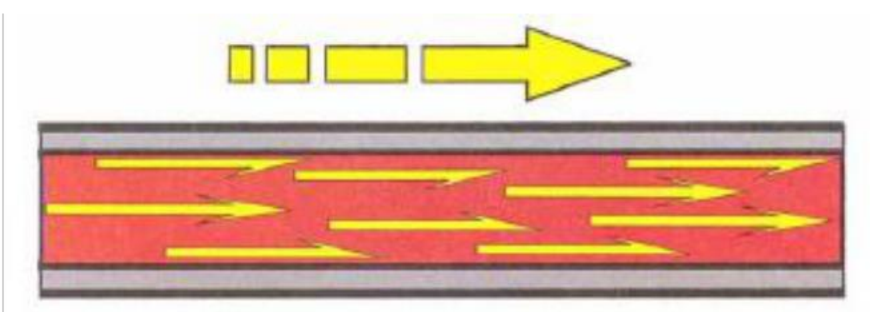


Figure 12: Laminar flow

As shown in Figure 12 at low velocity the flow is smooth, parallel layer of fluid that movie without mixing

1.8.2 TURBULENT FLOW

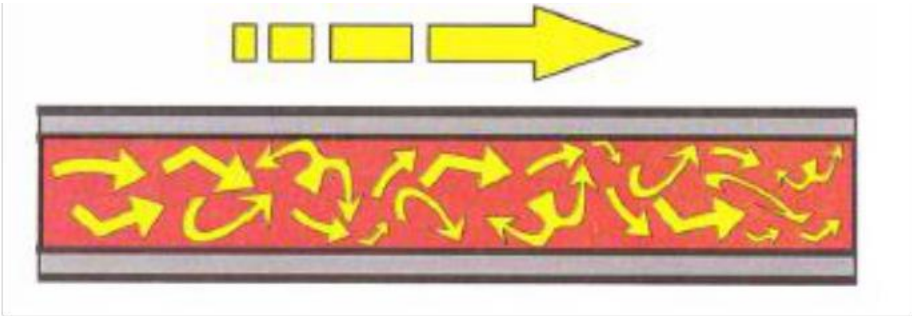


Figure 13: Turbulent flow

. As shown in Figure 13 at high velocity flow is not smooth layer of fluid is mixed it requires more energy to sustain

1.9 PRESSURE DROP

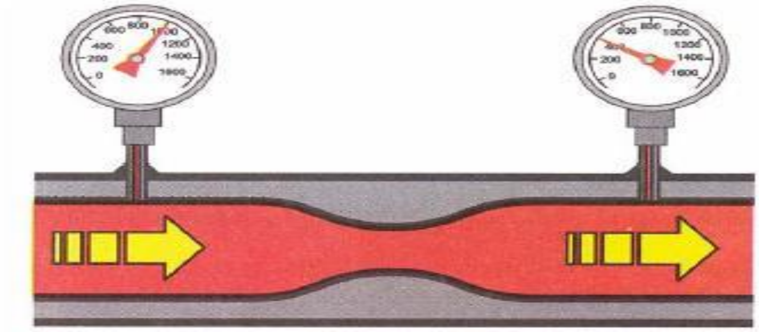


Figure 14: Flow past an orifice creates a pressure drop

As shown in Figure 14, when fluid moves through an office it experiences an energy loss. The pressure difference between upstream and downstream is known as a pressure drop, which results from both the flow and the restriction. The size of this pressure drop depends on the flow rate through the orifice, the orifice dimensions, and the fluids flow characteristics.

In Figure21, the downstream flow rate must equal the upstream flow rate because the fluid has no other path to exit. Even though the flow rates are the same, the lower downstream pressure means the fluid contains less energy

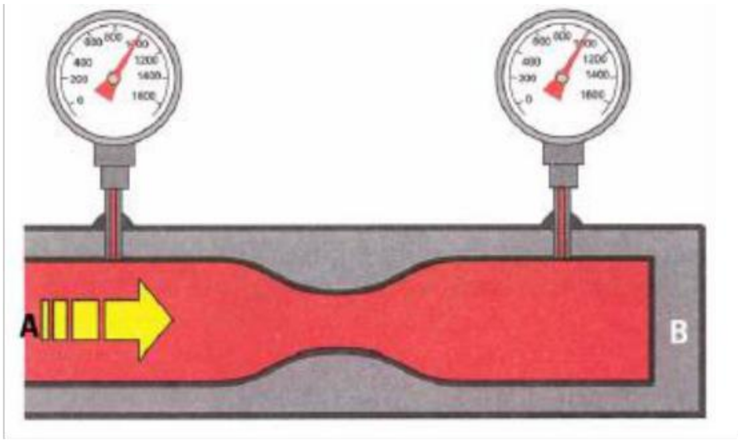


Figure 15: If here is no flow across an orifice, there is no pressure drop

When the pressure drop across a restriction depends on the flow rate through it, no flow means no pressure drop. Figure 15 illustrates this: with no flow through the orifice, the pressure is equal on both sides.

1.10 BERNOULLI'S PRINCIPLE

Bernoulli's principle states that for constant flow in a system, the sum of pressure and kinetic energy at different points remains the same. When a fluid moves through sections with varying diameters, as illustrated in Figure 16, the velocity must change accordingly. Bernoulli demonstrated that the pressure at point C must be lower than at points A and B due to its higher velocity at C corresponds to greater kinetic energy.

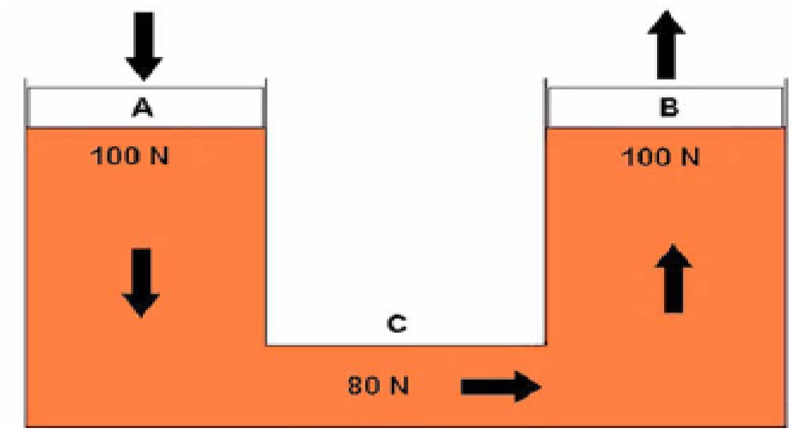


Figure 16: Velocity increase

Figure 17 shows the combined effect of velocity and friction change.

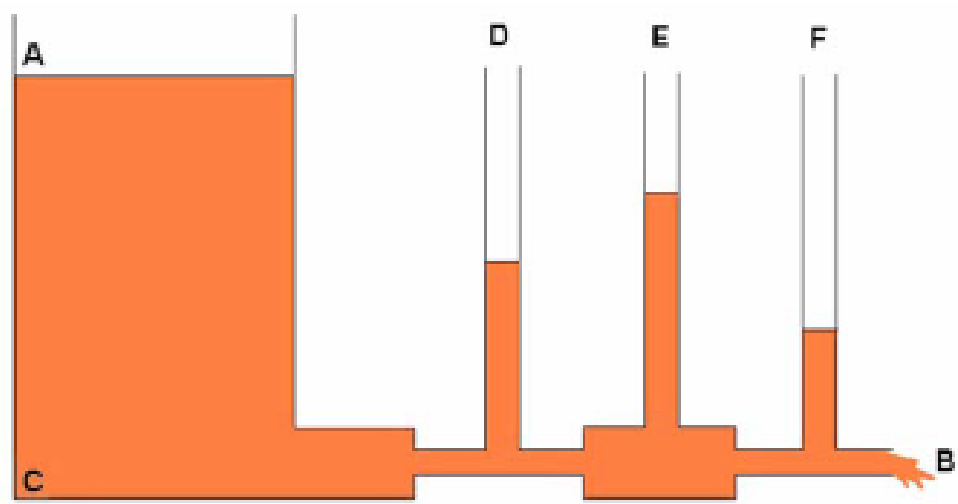


Figure 17: The combined effect of velocity and friction change.

Chapter-2

LITERATURE REVIEW

Dr. Prasad Bapat conducted study on designing special purpose machine. SPM'S(Special purpose machine) tools are developed and fabricated for special operations. This case study focuses on a manufacturing unit that uses various types of milling machines, including angular, horizontal, and vertical milling machine. The machine discussed in the study is intended for milling a 4 cylinder 40 HP engine block. The issue with the existing setup was that milling the locus clearance in the engine block on three separate machines was tedious and time – consuming. To address this, a spindle unit and the design were found to be safe.

K.Kishore Kumar, Dr. A. Srinath, M.Naveen and R.D.Pawan Kumar state that if the design does not meet the required standard, defects will occur in the final manufactured product. In design, even a small error can render the manufactured product useless, so more time should be devoted to designing a new product or modifying an existing one. Therefore, this design procedure can also be applied to manufacturing washers for M series bolts by modifying the punches and die plate.

S.P.SINHA AND P.D.MURARKA, this paper demonstrates that hydraulic presses are 3D complex structures, making precise analytical methods for stress and deformation analysis difficult and time intensive. based on this study, important guidelines for designing press frames have been derived.

Amol Rabade and his team members have conducted study on "Design and Development of SPM for face milling and two hole boring of a CR-22 block" These operations are not economical on conventional machines. The SPM was designed to achieve higher accuracy under desired conditions. The study concluded that the SPM was fabricated and commissioned satisfactorily. The SPM can complete 400 jobs per day. It also provides improved repeatability and accuracy due to precise automation, resulting in fewer rejections. Using the SPM reduces human interaction and drastically lowers workload, which helps decrease operator fatigue. This in turn reduces labour costs and increases the production rate.

Chapter -3

FINITE ELEMENT METHOD

3.1INTRODUCTION

The finite element method is numerical analysis technique used to find approximate solutions for a broad range of engineering problems. It began in the aerospace industry as a tool for analyzing stresses in complex aircraft structures. Developing from the matrix analysis method 1applied in aircraft design, the technique has become increasingly popular with researchers and practitioners.

3.2 STEPS INVOLVED IN FINITE ELEMENT ANALYSIS

The first step is finite element analysis is to model the given system. Two distinct approaches exist for modeling physical systems; the discrete lumped approach and the continuum approach, In the discrete lumped approach, the actual system is idealized as an assembly of elements, and equilibrium equations are written for each element. The resulting set of equations is then solved to obtain the solutions. In the continuum approach, the actual system is treated as a single continuum one or more equilibrium equations are written and solved to determine the system response.

When a continuum is divided into finite elements, it is referred to as a finite element model. A typical finite element model consists of nodes, elements, material properties, degrees of freedom, boundary conditions, externally applied loads, and the type of analysis. Engineers and designers must conduct the model carefully to ensure that the FEA results are relevant. The results are determined entirely by the model

In practice, practitioners must decide on the mesh layout, specifically the number of nodes and elements. Areas where abrupt changes in the unknown variable are expected, such as stress concentrations around holes, need a higher density of nodes and elements compared to regions with gradual changes. This is known as gradation. After selecting the mesh layout, practitioners must choose the type of analysis (static /dynamic , linear/ nonlinear, beam theory, plane stress, three-dimensional, etc.),the type and number of degrees of freedom at each node(deflection, rotation, temperature, flux, etc.), boundary conditions, relevant element information (type, number of nodes per element, etc.),material properties, damping, and external loads at the nodes.

After the finite element model is established by selecting all parameters for the corresponding mesh, it must be entered into the FEA code for execution. An examination of leading commercial FEA software shows that users are required to supply the finite model data through a data file. This file typically contains five main sections: control, nodal, element, material, and loading. While the last four sections are self–explanatory, the control section contains details like the problem description (heading), total node count, type of analysis, and similar information.

These codes have the six modules.

NODE MODULE: Node module generates nodal co-ordinates and degree of freedom. BOUNDARY CONDITION

MODULE: It various element types in an element library, evaluates

element matrices and assemblies these matrices.

LOADING MODULE: It applies the externally applied loads to the proper nodes and elements. SOLUTION MODULE:

This module solves the system of equation that results from the assembly

process of the element module for the unknown modules.

FEA codes consist of pre processor and post processor. Both are integral to the FEA process. Pre-processing occurs before the analysis and helps the user automate and manage the input required by the initial modules. Post –processing takes place after the analysis and allows the user to visualize the results in various graphical formats for better understanding and interpretation. The preceding paragraph described the procedure used to perform finite element analysis of a system.

The procedure followed to obtain the finite element analysis of a system has been described. We will now discuss some of the involved steps in detail.

3.3 MODELING:

The initial step is to model the object in a CAD system. There are several commercially available software packages that help users create models of physical systems. However, they require extensive training and practice. These packages have undergone many improvements since they were first introduced, AutoCAD, SOLIDWORKS, MECHANICAL DESKTOP, ANSYS, 1-IDEAS, ETC. are some of the popular commercially available CAD software. Some of these also permit analysis of the model while others are only meant for modeling.

3.4 ELEMENT SHAPES

Discretizing or dividing a continuum u into smaller parts requires engineering judgment. The initial step in this process involves choosing the shape or configuration of the fundamental element that will be used for the analysis.

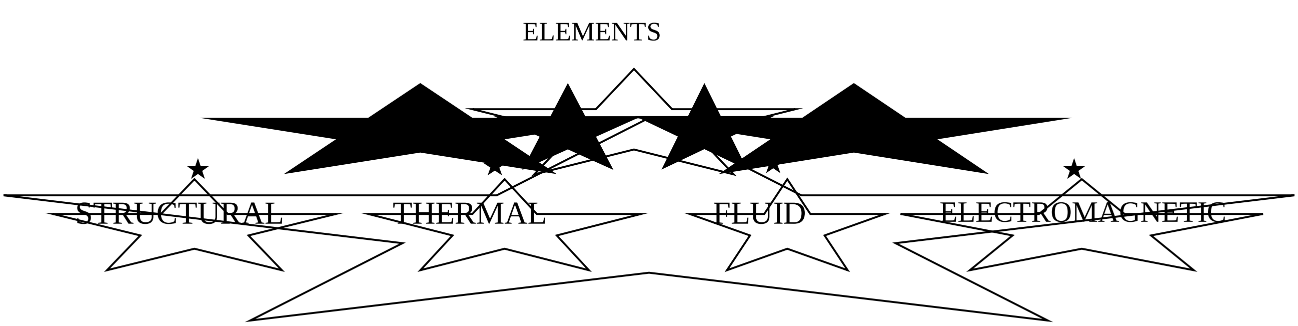


Figure 18: Element shapes

After selecting the element type, the entire continuum is discretized using that element. This process is known as mesh generation.

3.5 MESH GENERATION

Element meshes were created

Manually In manual mesh generation, the analyst discretize the simplified geometry of the object under study by converting the geometric model into nodes and elements. Nodes are defined by specifying their coordinates, while element connectivity determines the elements. Manual meshing is inefficient and error-prone, and the meshing date can quickly become large and difficult to complex objects, particularly three dimensional ones.

Due to the widespread adoption of computer graphics and CAD technology, efforts to automate mesh generation have increased. Numerous algorithms, schemes, and methods exist for creating meshes, each offering different degrees of automation and requiring varying amounts of automation and requiring varying amounts of user input. This section categorizes mesh generation as either semiautomatic or fully automatic. Fully automatic mesh generation refers to a method where the only required inputs are the geometry and mesh attributes, such as mesh density, element type, boundary conditions, and input, like subdividing the object into sub domains or regions, is considered semiautomatic

3.6 SEMIAUTOMATIC METHOD:

These approaches are also called “interactive mesh generation methods” to highlight their key characteristic: the analyst must interact with the mesh generator to produce the mesh generator to produce the mesh. Semiautomatic mesh generation methods fall into two group: and the solid– modeling based group. Within each group, methods are further categorized as either node based approaches or region based approaches.

3.7 FULLY AUTOMATIC METHODS:

These methods are mainly developed using solid modeling theory to enable complete automation, and they function exclusively with solid models. A fully automatic mesh generator is typically activated at the user level through a command like “mesh solid attd”, where“att”denotes mesh attributes and“d”is a digitizer used to select the solid for meshing. This means that in mesh automation, user input is restricted to defining the solid and specifying mesh density parameters.

3.8 TYPES OF ANALYSIS:

The user must determine which analysis type to run on the finite element model, This decision depends on factors such as the required result accuracy, material type, applied load, analysis types available, including static/ dynamic, linear/nonlinear, thermal, fluid flow, acoustic and more. These can also be combined, for example static-linear or static –nonlinear. Among these, static analysis is the most widely used and popular option.

3.9 DEFINITION OF STATIC ANALYSIS:

Static analysis determines the effects of steady loading conditions on a structure while neglecting inertial and damping effects. To analyze an object, the loads acting on the system must be loads acting on the system must be defined. The object's boundary conditions must also be provided to the software solving the problem. Acting loads include forces and their nature, such as shear, tensile, compressive, or bending, and whether forces are in-plane or eccentric must be specified. Boundary conditions cover the state of the continuum's boundary, including whether it is fixed, permitted to translate or rotate, allowable boundary deflections, temperature, and other factor. The accuracy of the results depends on how accurately the applied loads and boundary conditions are specified.

Material properties must be defined before analyzing the object. While some software include as database with properties for several commonly used materials, the properties of any other materials, the properties of any other materials being used must b specified manually. Properties that typically need to be specified include modulus of elasticity, Poisson's ratio, ultimate tensile strength, shear strength, coefficient of thermal expansion, emissivity, and others.

The software requires data to be entered in a specific format. With user input, preprocessors can automatically create these data files. This means the user does not have to be familiar with the precise structure of the required data file. Once the user builds a finite element model using a given.

CHAPTER -4

DESIGN OF COMPONENT

4.1 DESIGN OF TIE ROD

D= Diameter of Rod =50 mm [1]

F= Force Acting on Each Tie rod = $\frac{10T}{2}$

$= \frac{10\times1000\times9.81}{2} = 49050N$

A= $\frac{\pi}{4}(D)2 = \frac{\pi}{4}\times502$ = 1962.5 mm^2

Stress = $\frac{F}{A} = \frac{49050}{1962.2}$ = 24.99 N/mm^2

4.2 DESIGN OF 10T HYDRAULIC CYLINDER

Material used-Mild Steel F=

Required Force is 10T

= 10×1000×9.81 =98100 N

Working pressure = Pr = 15 N/mm^2 ID =

Internal diameter = 100 mm OD= Outside

diameter = 120 mm

T = Thickness = $\frac{OD-ID}{2} = \frac{120-100}{2}$ =10mm

Longitudinal Stress = $\frac{Pr\times ID}{4T}$

$= \frac{15\times100}{4\times10} = \frac{1500}{40}$

Stress = 37.5 N/mm

Hoop Stress = $\frac{PD}{2T} = \frac{15\times100}{2\times10}$

$= \frac{1500}{20} = 75N/mm^2$

4.3 DESIGN OF FIXED FRAME

P= 98100 N L= 750 mm

We have = $\frac{M}{I} = \frac{\sigma b}{4} = \frac{E}{R}$ [2]

M= Bending Moment in N-mm

M = $\frac{P\times L}{4} = \frac{98100\times550}{4} = 13488750$N-mm

I = $\frac{b\times h3}{12} = \frac{500\times(80)3}{12} = 21333333.3$ mm^4

Z= Shear Modulus = $\frac{I}{y}$ = mm^3

Y = $\frac{80}{2} = 40$ mm

Z= $\frac{21333333.3}{40}$ = 533333.3 mm^3

Bending Stress = σb $= \frac{M}{Z}$

$= \sigma b = \frac{13488750}{533333.3} = 25.29$N/mm

4.4 DESIGN OF MOVING FRAME.

P= 98100 N l=550 mm

We have $\frac{M}{I} = \frac{\sigma b}{4} = \frac{E}{R}$

M = Bending Moment in N-mm

M = $\frac{P\times L}{4} = \frac{98100\times550}{4} = 13488750$N-mm

I = Moment of Inertia in mm^4

I = $\frac{b\times h3}{12}$ = $\frac{500\times(30)3}{12}$ =1125000mm4

Y = $\frac{30}{2}$ =15 mm

Z= Shear Modulus = $\frac{I}{y}$ = $\frac{1125000}{15}$ =75000 mm^3

Bending stress = $\sigma b = \frac{M}{Z}$ =N/ mm^2

Bending stress= $\frac{1348875}{0\ 75000}$ = 179.85 N/mm^2

4.5 DESIGN OF MODIFIED MOVING FRAME.

P= 98100 N l=550 mm

We have $\frac{M}{I} = \frac{\sigma b}{4} = \frac{E}{R}$

M = Bending Moment in N-mm

M = $\frac{P\times L}{4}$ = $\frac{98100\times550}{4}$ =13488750N-mm

I = Moment of Inertia in mm^4

I = $\frac{b\times h3}{12}$ = $\frac{500\times(60)3}{12}$ = 9000000mm4

Y = $\frac{60}{2}$ =30 mm

Z= Shear Modulus = $\frac{I}{y}$ = $\frac{9000000}{30}$ = 300000 mm^3

Bending stress = $\sigma b = \frac{M}{Z}$ =N/ mm^2

Bending stress= $\frac{1348875}{0\ 30000}$ = 44.96 N/mm^2

CHAPTER -5

DESIGN AND DEVELOPMENT OF 10MT VALVE TESTING EQUIPMENT. 5.12D MODEL

ASSEMBLY VIEW OF10MT VALVETESTING EQUIPMENT.

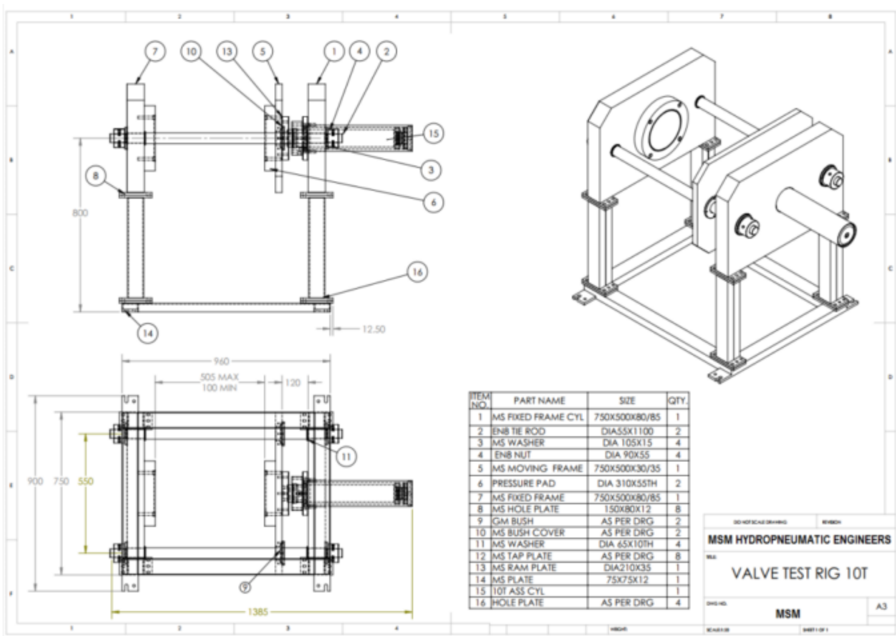


Figure 19: 2D Model assembly view of 10MT valve testing equipment.

5.23D MODEL ASSEMBLY VIEW OF 10MT VALVE TESTING EQUIPMENT.

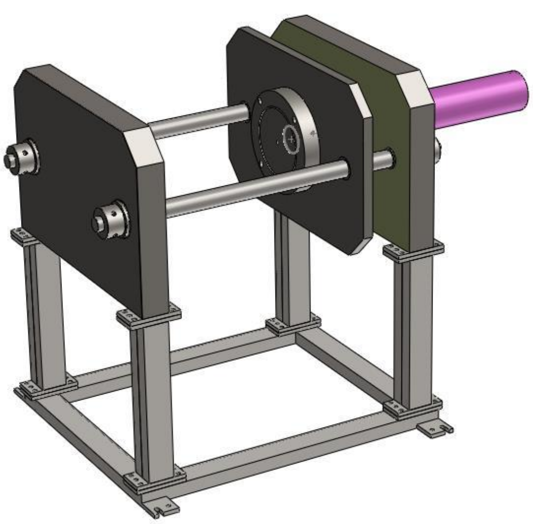


Figure 20: 3D Model assembly view of 10MT valve testing equipment.

5.32D MODEL VIEW OF TIE ROD.

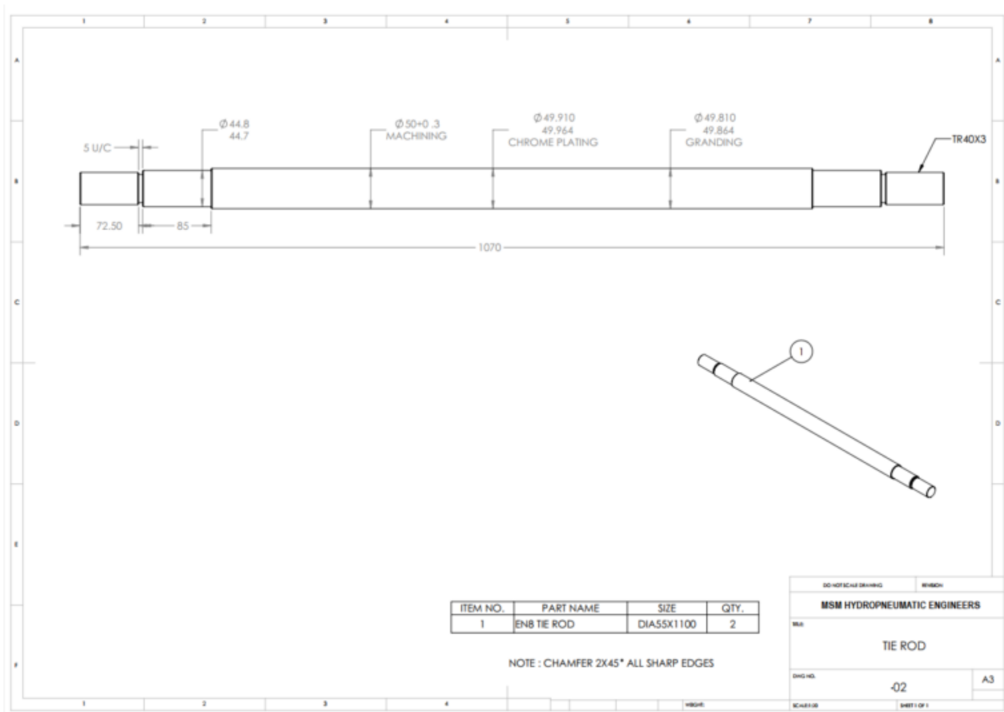


Figure 21: 2D Model view of the tie rod.

5.4 3D MODEL VIEW OF TIE ROD.

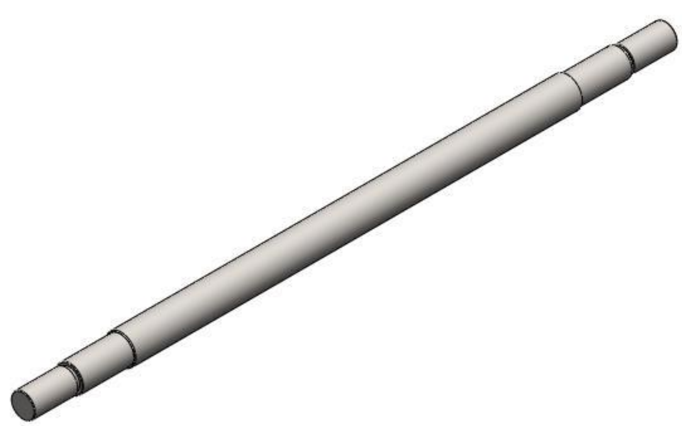


Figure 22: 3D Model view of tie rod.

5.5 2D MODEL VIEW OF MOVING FRAME.

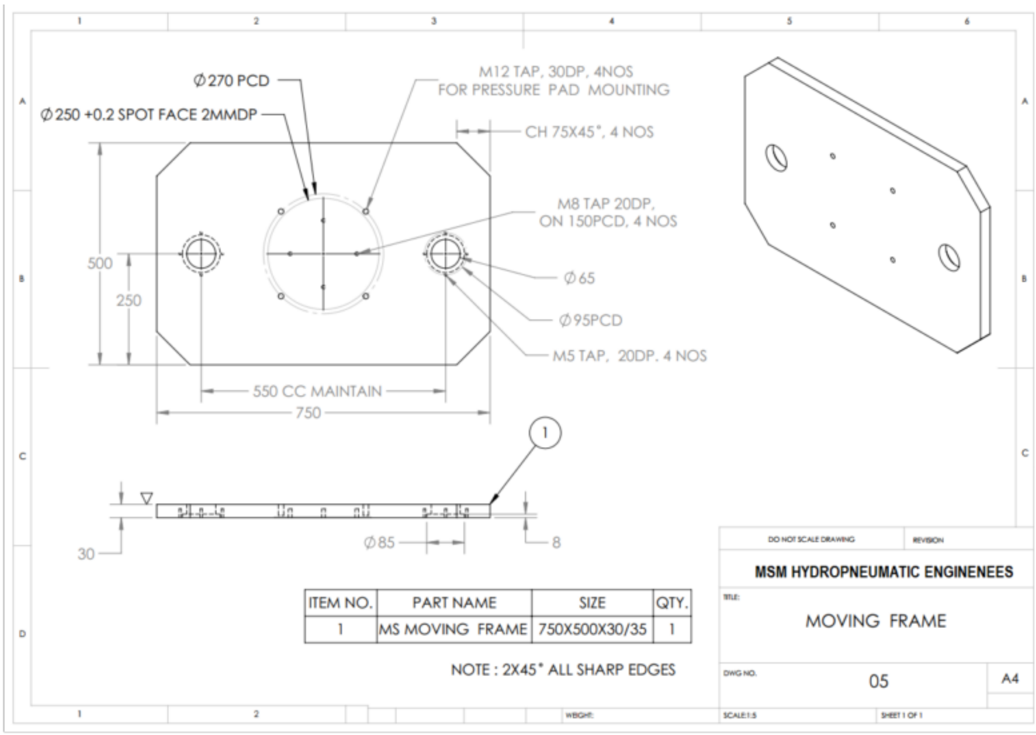


Figure23: 2D Model view of moving frame.

5.63D MODEL VIEW OF MOVING FRAME.

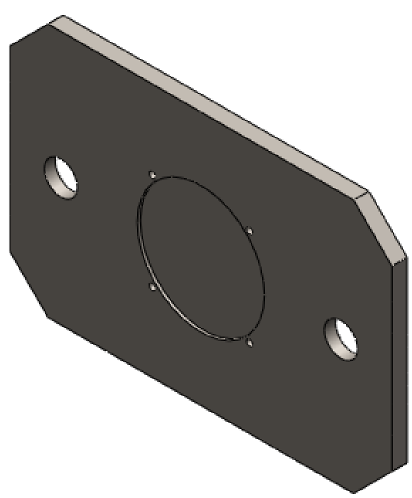


Figure 24: 3D Model view of moving frame.

5.72D MODEL VIEW OF FIXED FRAME.

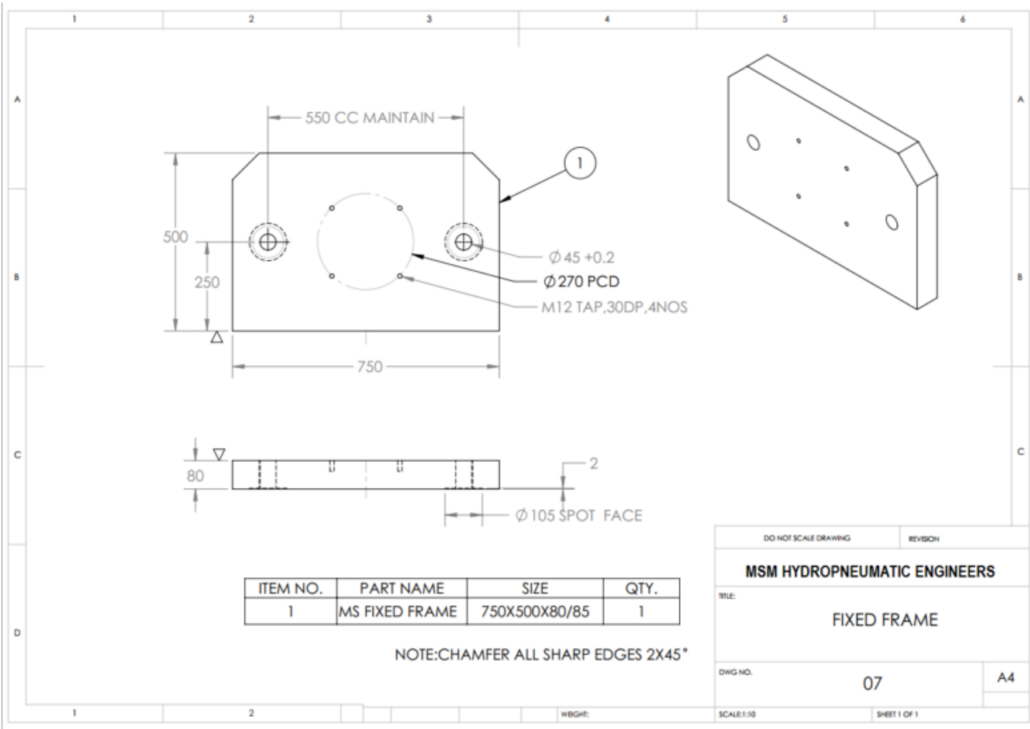


Figure 25: 2D Model view of fixed frame.

5.83D MODEL VIEW OF FIXED FRAME.

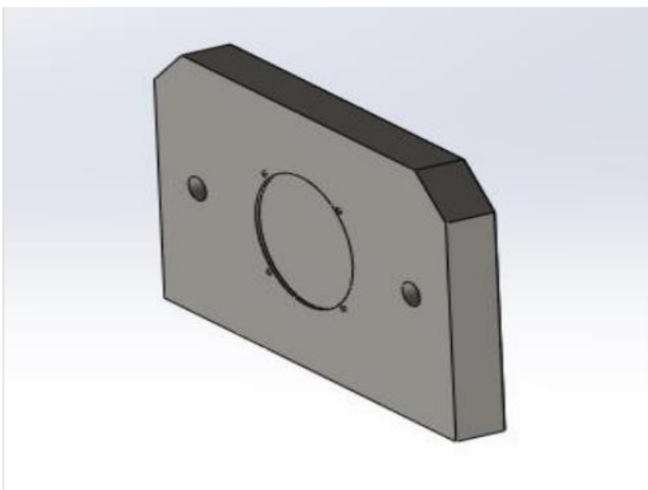


Figure 26: 3D Model view of fixed frame.

5.9 2D MODEL VIEW OF CYLINDER.

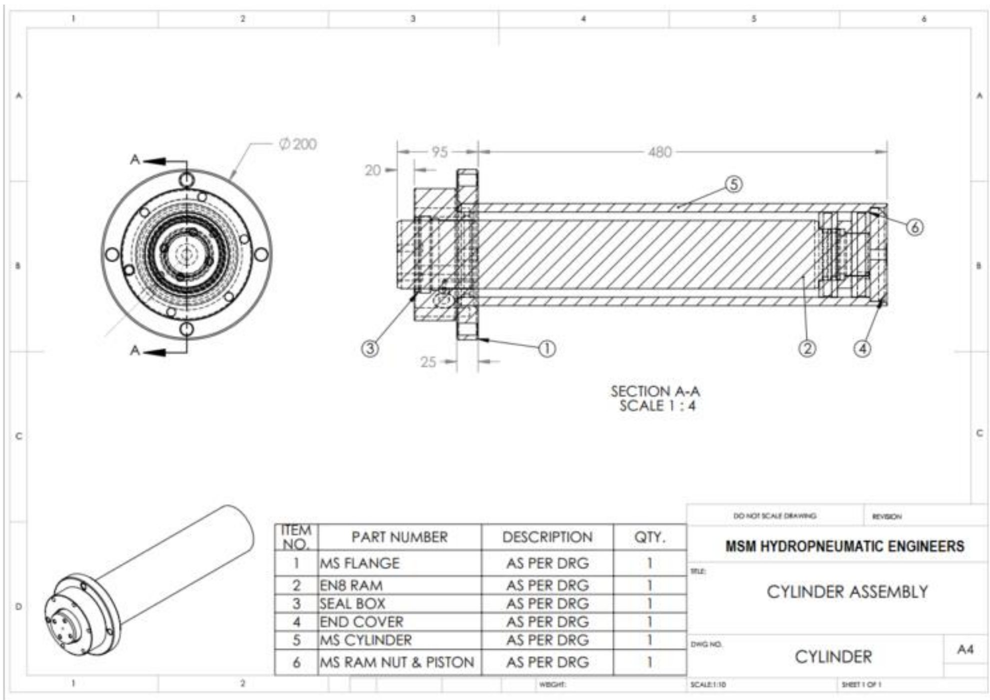


Figure 27: 2D Model view of cylinder

5.103D MODEL ASSEMBLY VIEW OF CYLINDER.

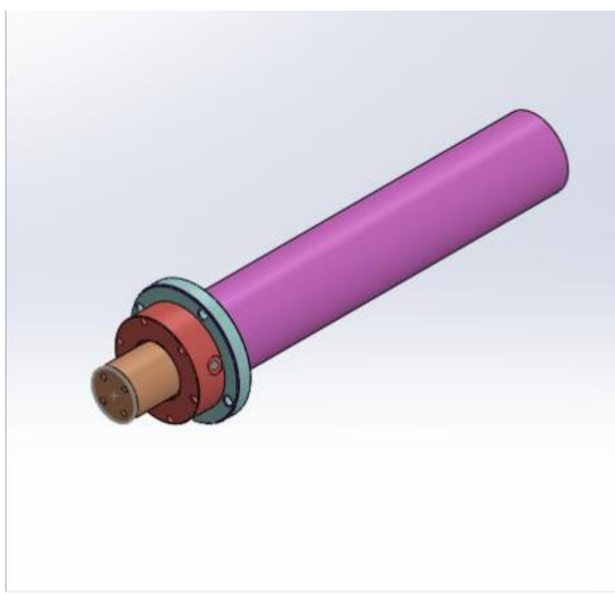


Figure 28: 3D Model assembly view of cylinder

5.11 2D MODEL VIEW OF FIXED FRAME STAND.

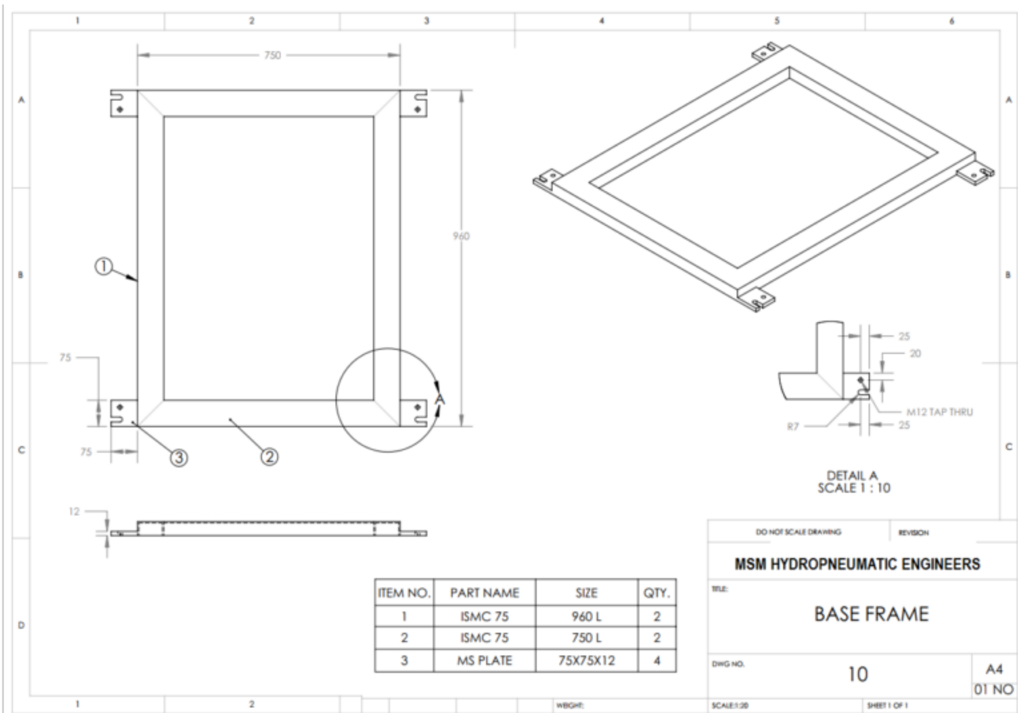


Figure 29: 2D Model view of base frame.

5.12 3D MODEL VIEW OF FIXED FRAME.

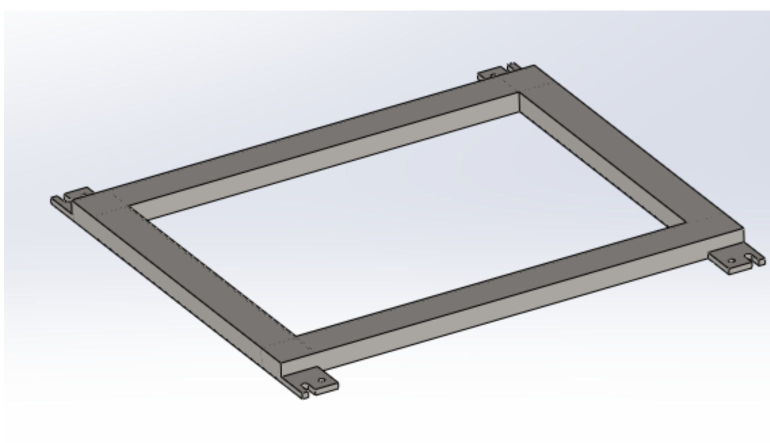


Figure 30: 3D Model view of base frame.

5.132D MODEL VIEW OF NUT.

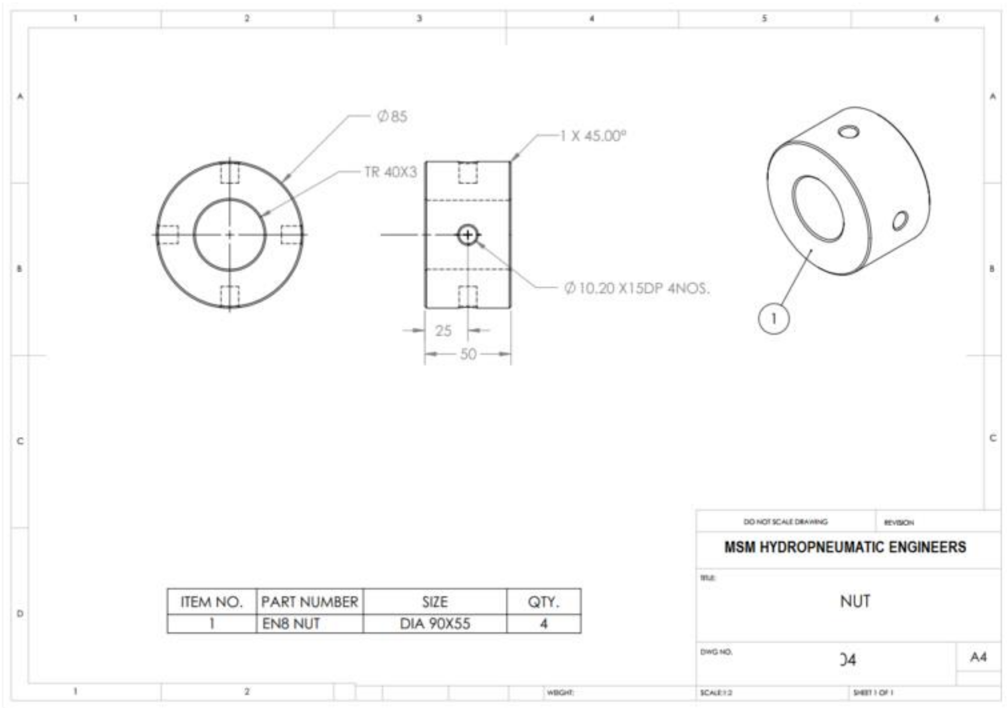


Figure 31: 2D Model view of nut.

5.14 3D MODEL VIEW OF NUT.

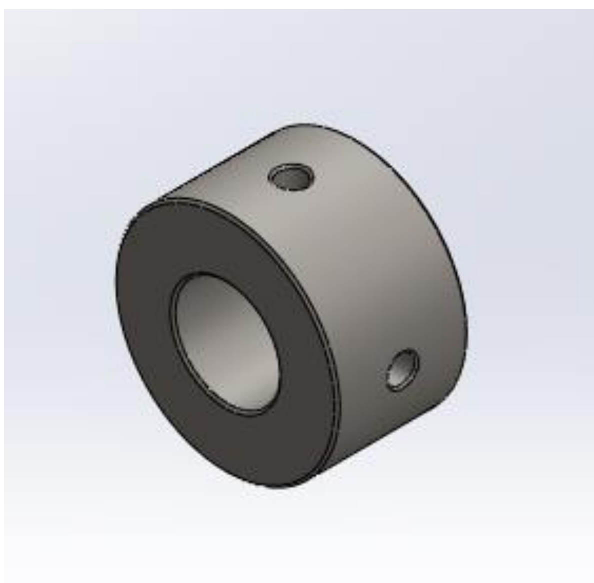


Figure 32: 3D Model view of nut.

5.15 2D MODEL VIEW PRESSURE PAD.

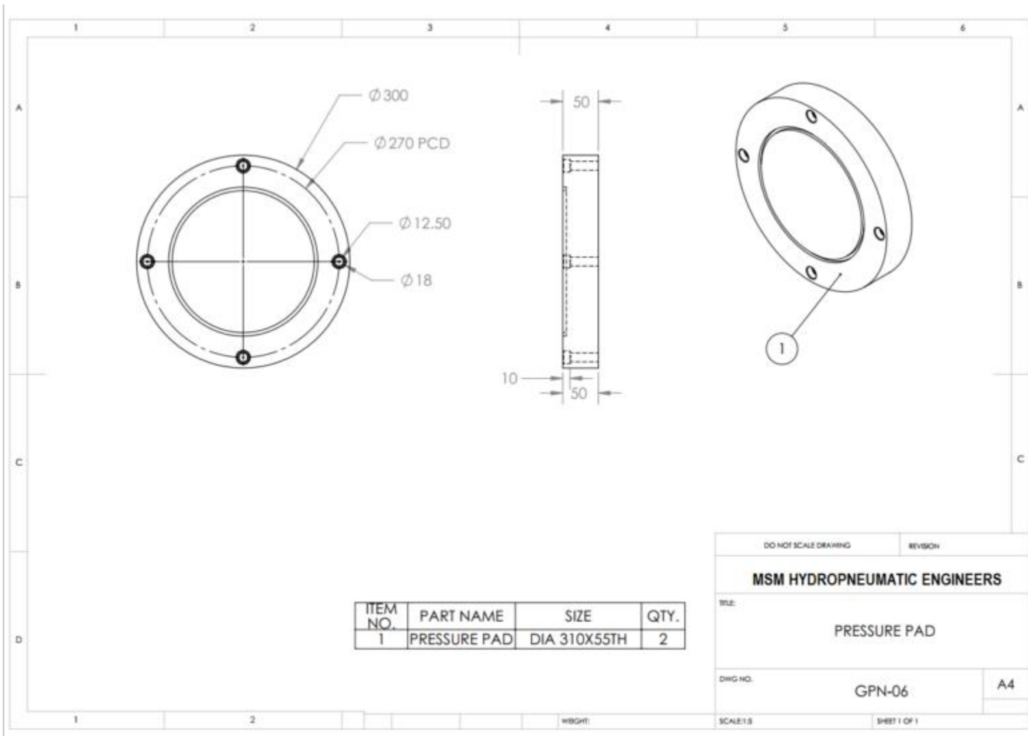


Figure 33: 2D Model view pressure pad.

5.16 3D MODEL VIEW PRESSURE PAD.

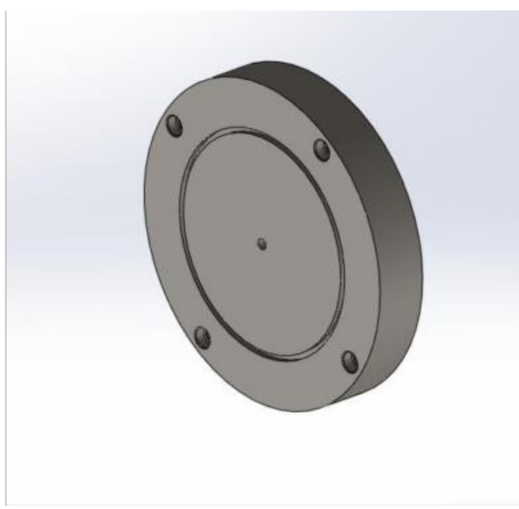


Figure 34: 3D Model view pressure pad.

5.17 2D MODEL VIEW OF RAM PLATE.

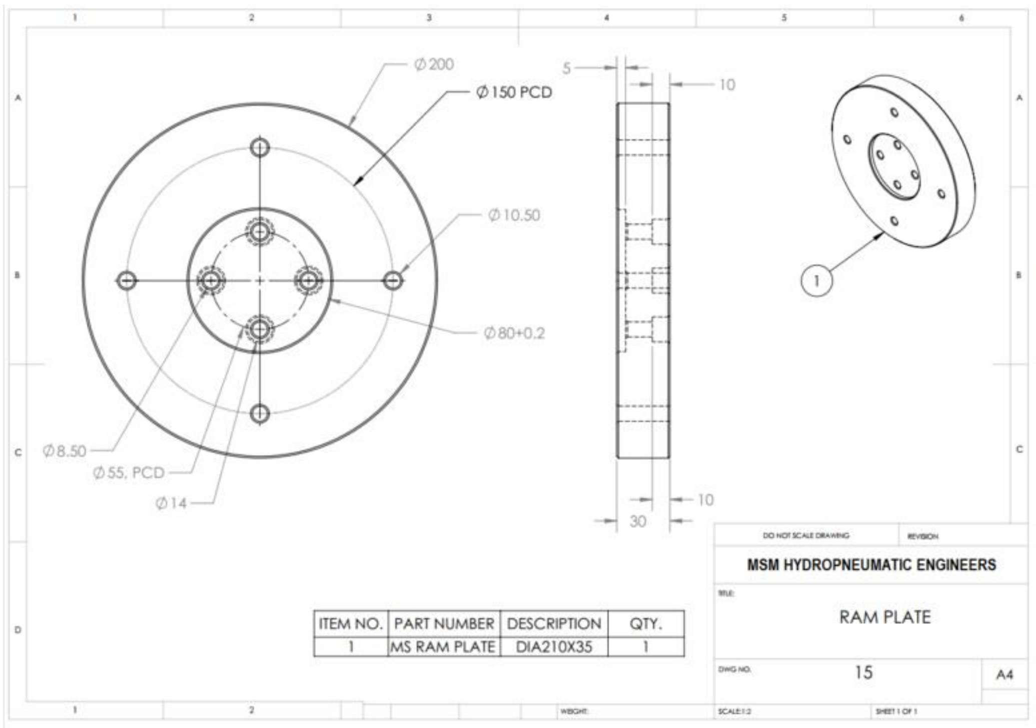


Figure 35: 2D Model view of ram plate.

5.18 3D MODEL VIEW OF RAM PLATE.

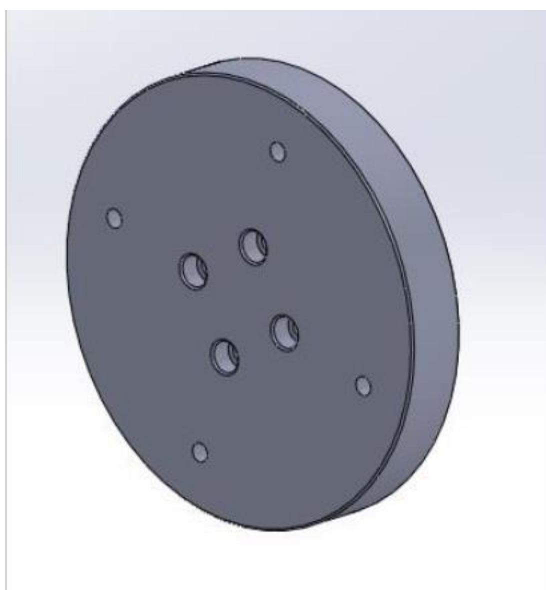


Figure 36: 3D Model view of ram plate.

5.19 2D MODEL VIEW GM BUSH.

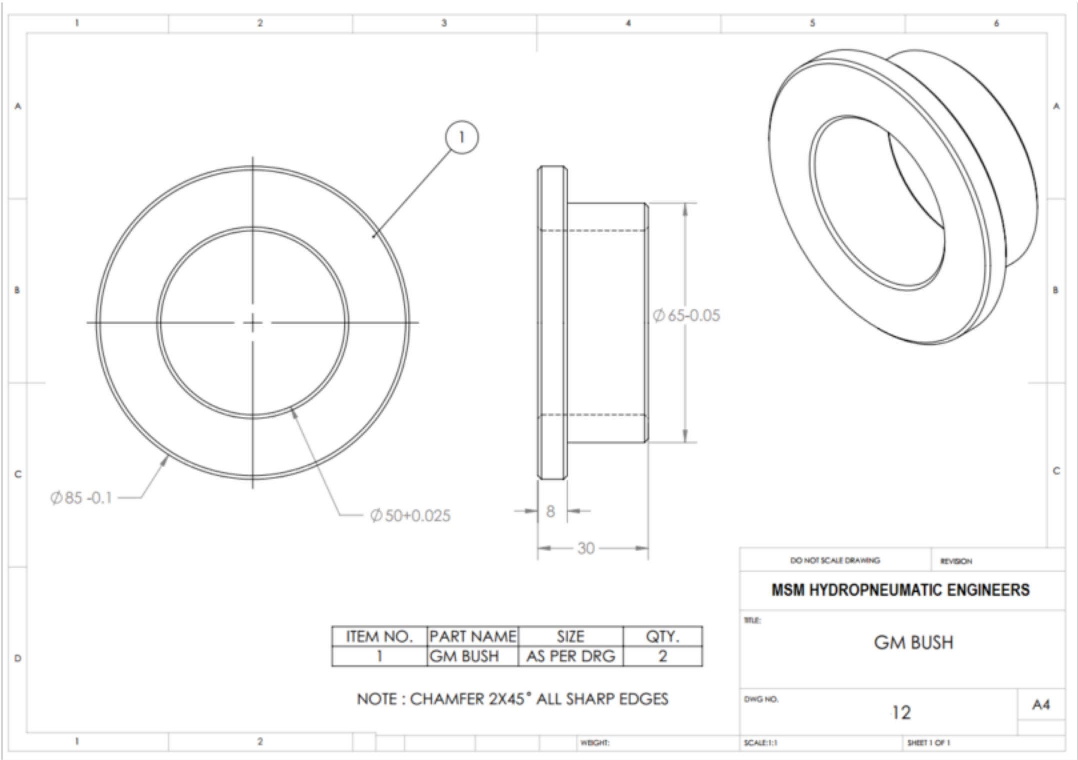


Figure 37: 2D Model view gm bush.

5.20 3D MODEL VIEW GM BUSH.

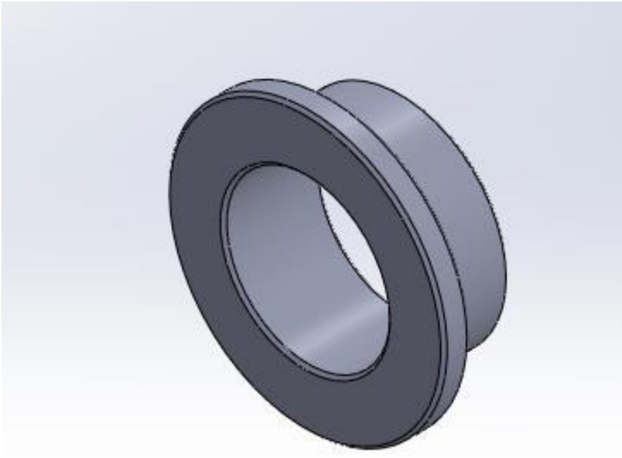


Figure 38: 3D Model view gm bush.

Chapter 6

Finite Element Analysis of Valve Testing Equipment

6.13D Geometric Model View of 10MT on Hydraulic Cylinder

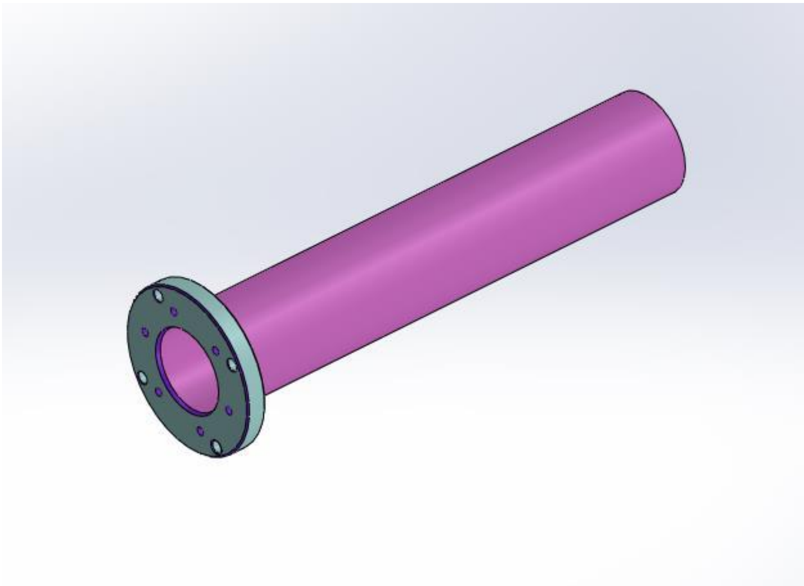
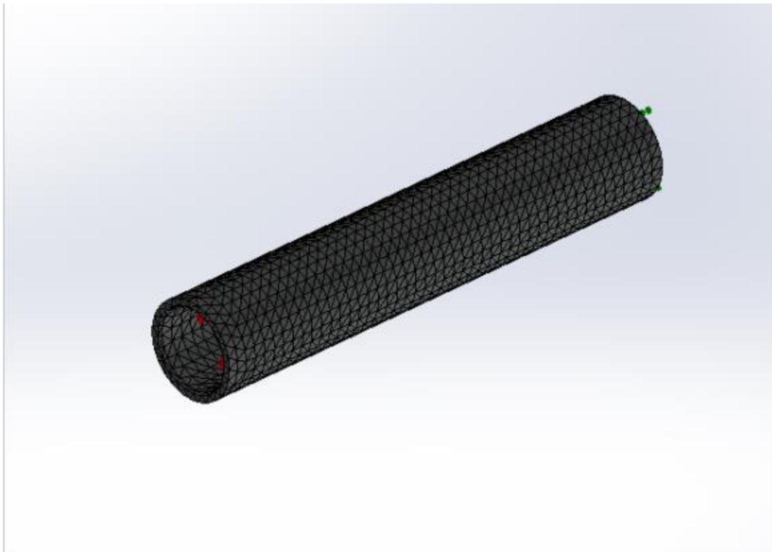


Figure 39 3D Geometric model view of 10MT on hydraulic cylinder

6.2Meshed View of 10MT of HydraulicCylinder



6.3Boundary Conditions– Pressure Applied 15Mpa and Fixed Face

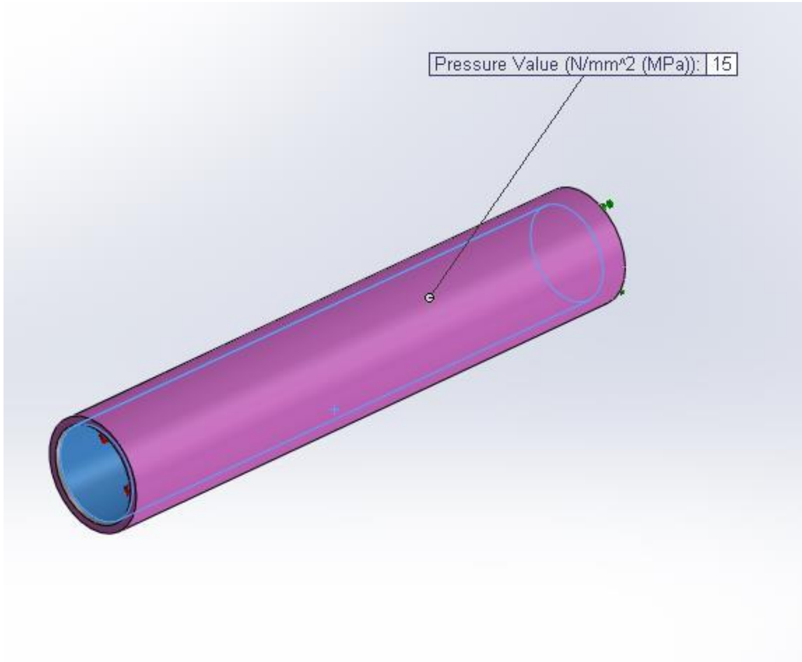


Figure 41 Boundary conditions –pressure applied 15 MPa and fixed face

6.4 Stress Induced in 10MT on Hydraulic oil Cylinder

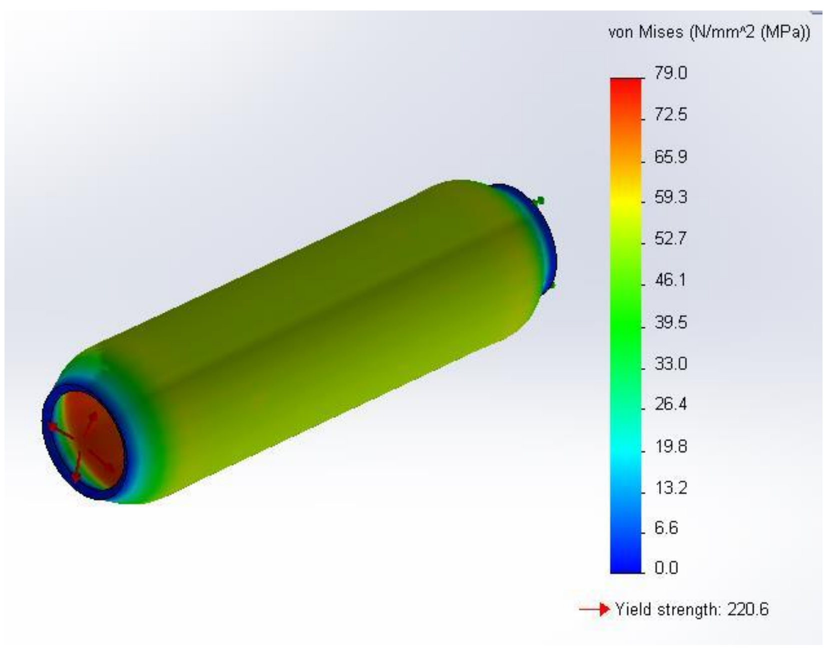


Figure 42 Stress 79MPa induced in 10MT hydraulic cylinder

6.5Deformation Induced in 10MT Hydraulic oil Cylinder-

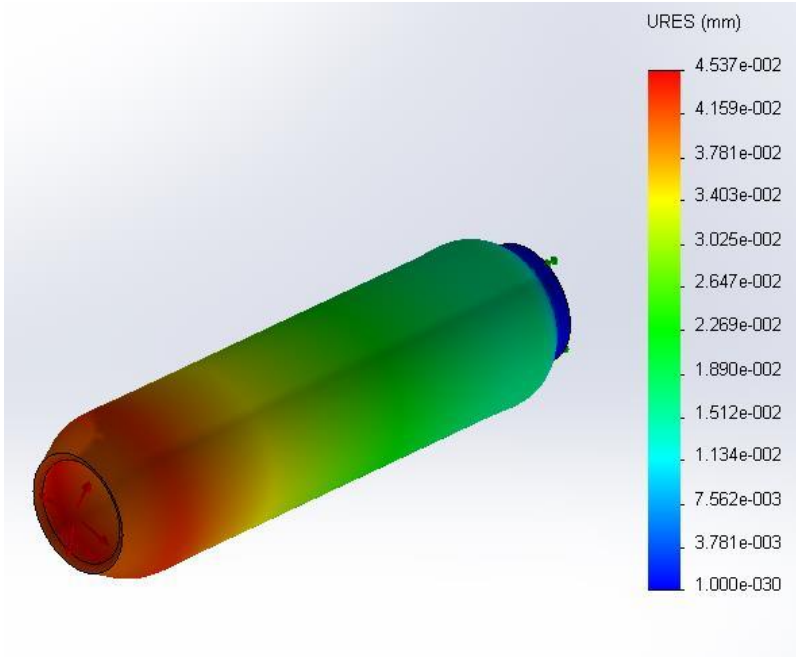


Figure 43 Deformation induced in $4.5376e^{-002}$ 10MT hydraulic cylinders

6.63D Geometric Model View of Tie Rod.

Figure 44 3D Geometric model view of tie rod

6.7Meshed View of Tie Rod.

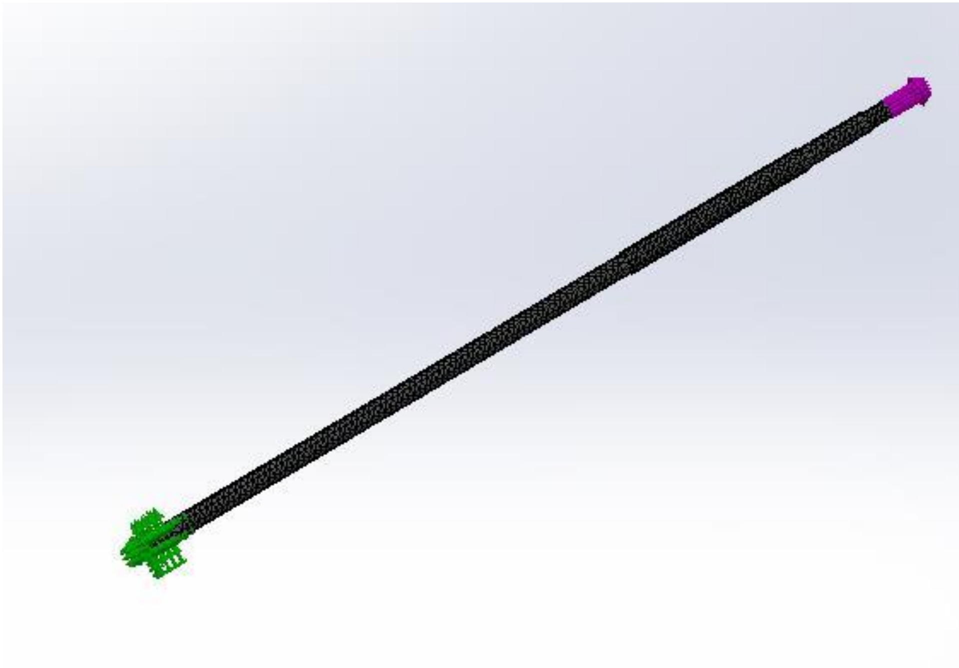


Figure45 Meshed view of tie rod

6.8 Boundary Conditions– Pressure Applied Force 49050N and Fixed Face

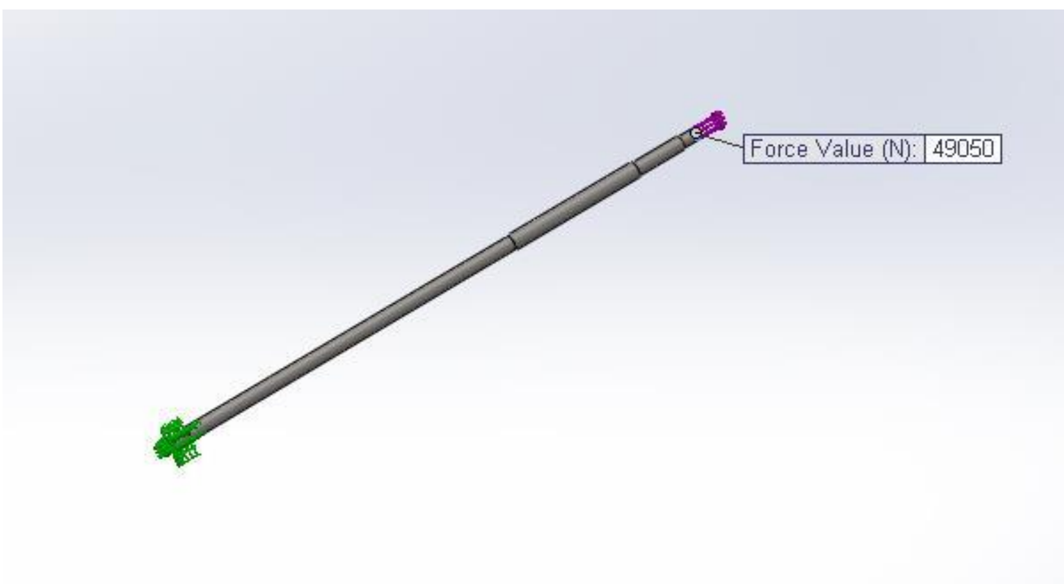


Figure 46 Boundary conditions –force applied 49050N and fixed face

6.9Stress Induced In Tie Rod

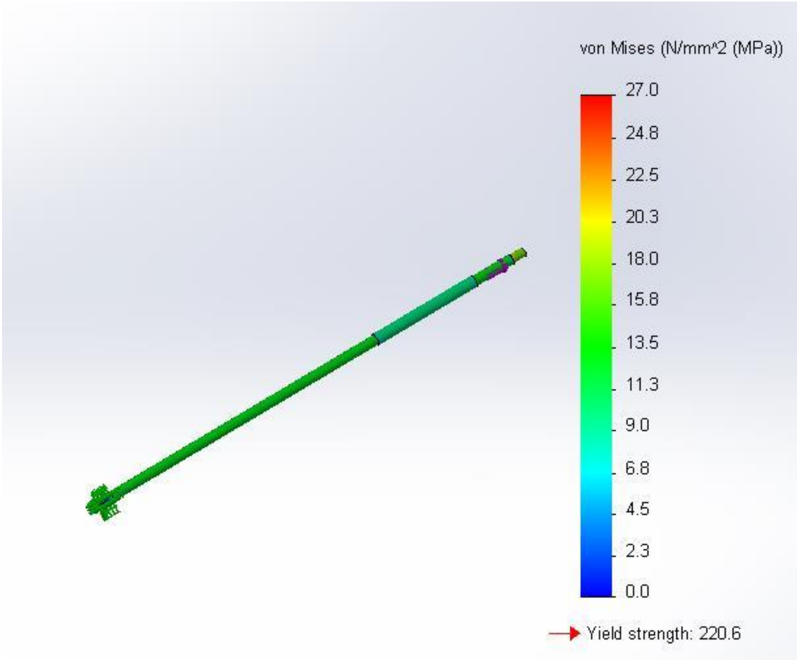


Figure 47 Stress of 27MPa induced in tie rod

6.10Deformation Induced in Tie Rod

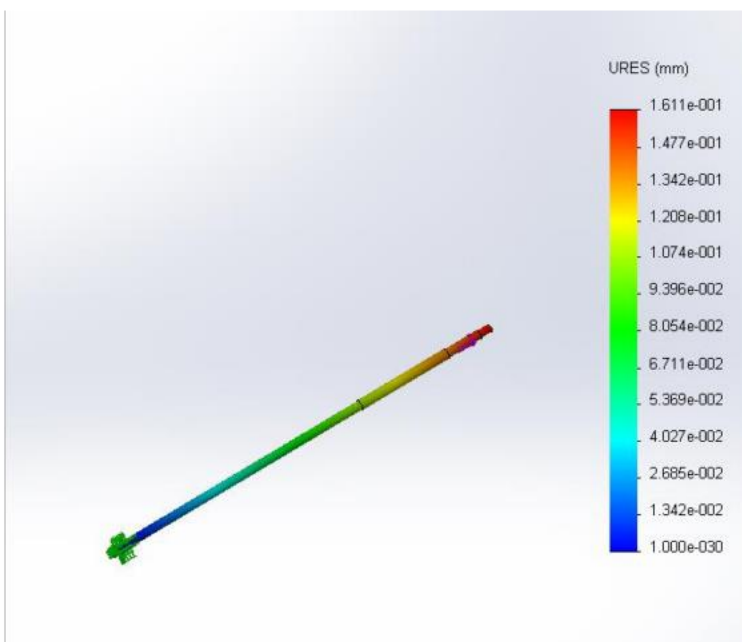


Figure 48 Deformation of 1.611e [illegible] mm induced in tie rod

6.113D Geometric Model View of Moving Frame

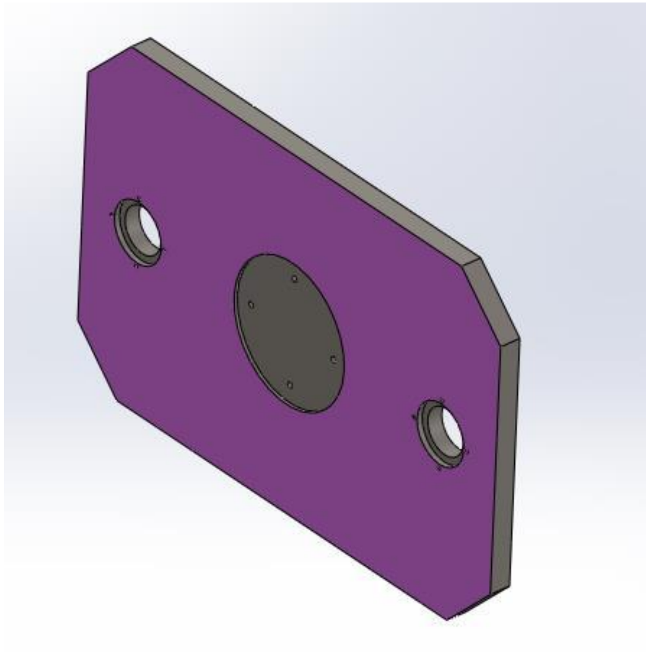


Figure 49 3D Geometric model view of moving frame

6.12Meshed View of moving frame

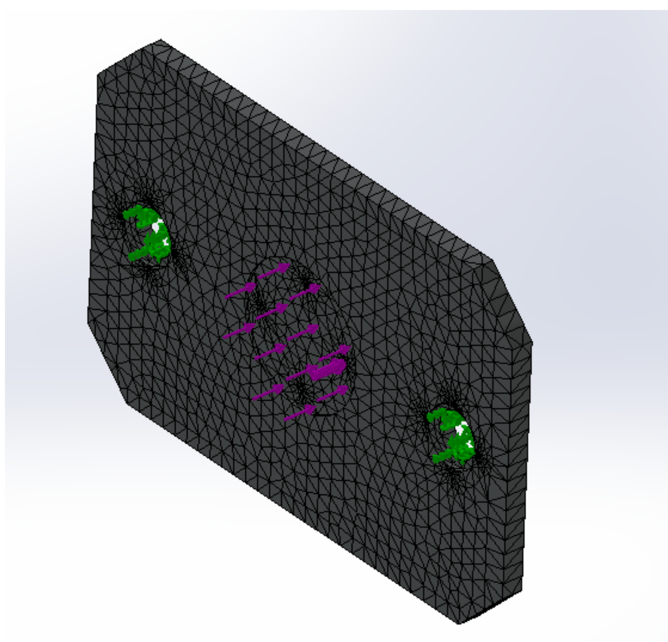


Figure 50 Meshed view of moving frame

6.13Boundary Conditions– Force Applied and Fixed At the Face

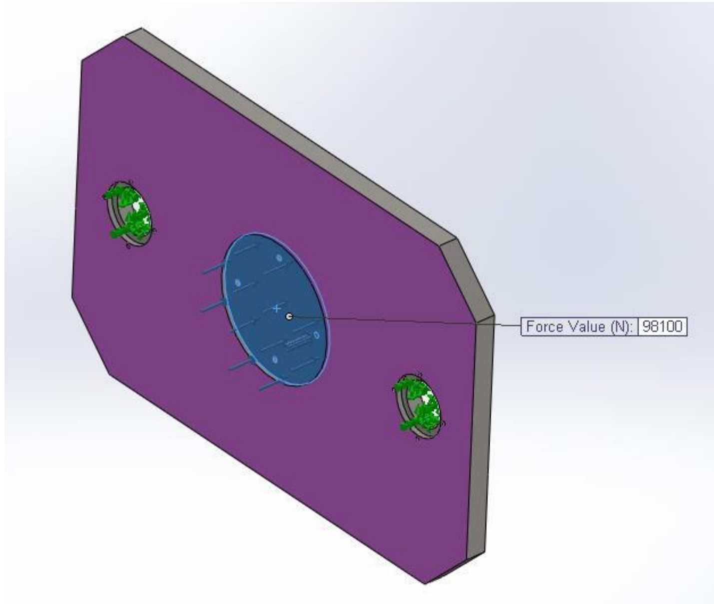
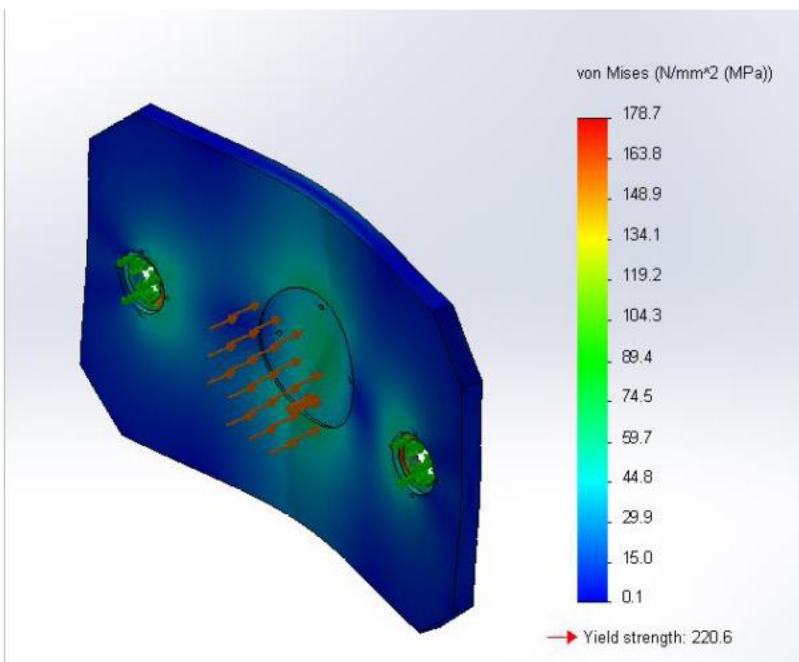


Figure 51 Boundary conditions –force 98100N applied and fixed at the face

6.14Stress Induced in Moving Frame



6.15Deformation Induced in Moving frame

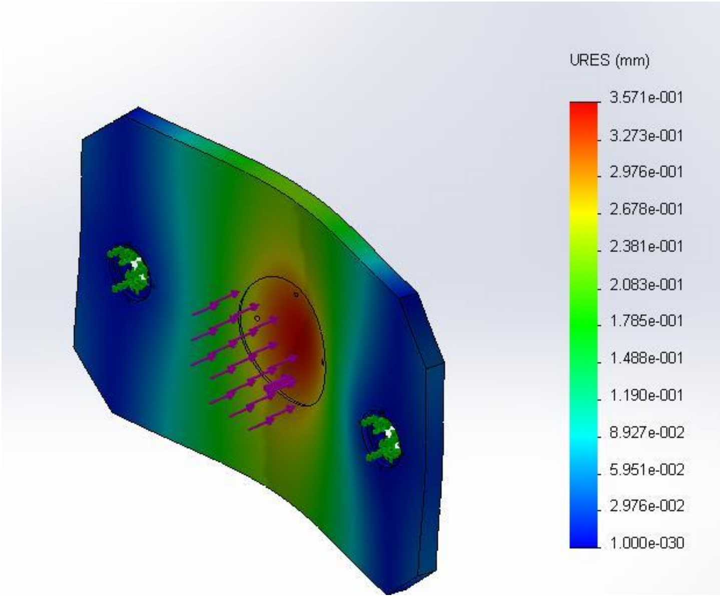


Figure 53 Deformation3.5e -001mm induced in moving frame

6.16 3D Geometric Model View of Fixed Frame

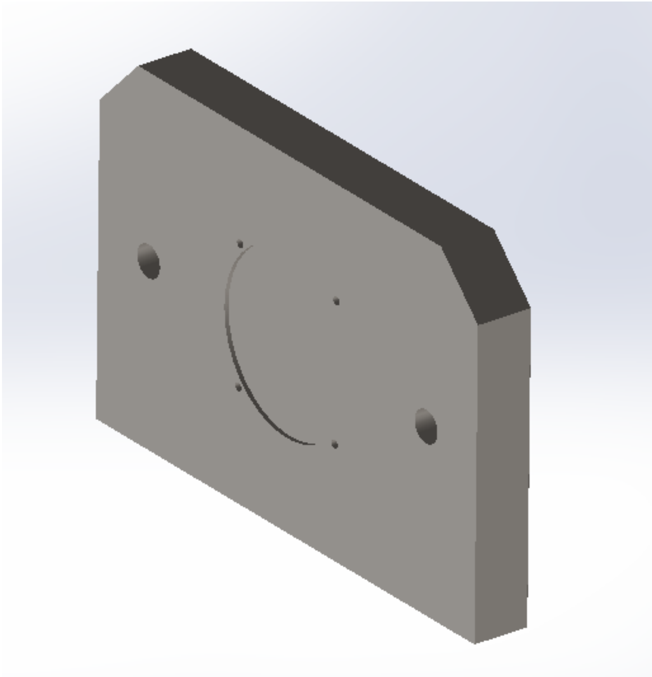


Figure 54 3D Geometric model view of fixed frame

6.17 Meshed View of fixed frame

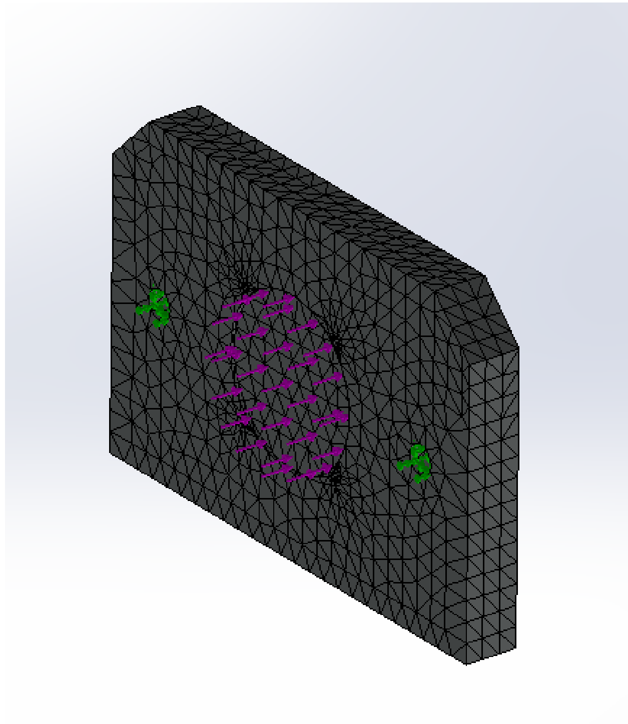


Figure 55 Meshed view of fixed frame

6.18 Boundary Conditions– Force Applied and Fixed At the Face

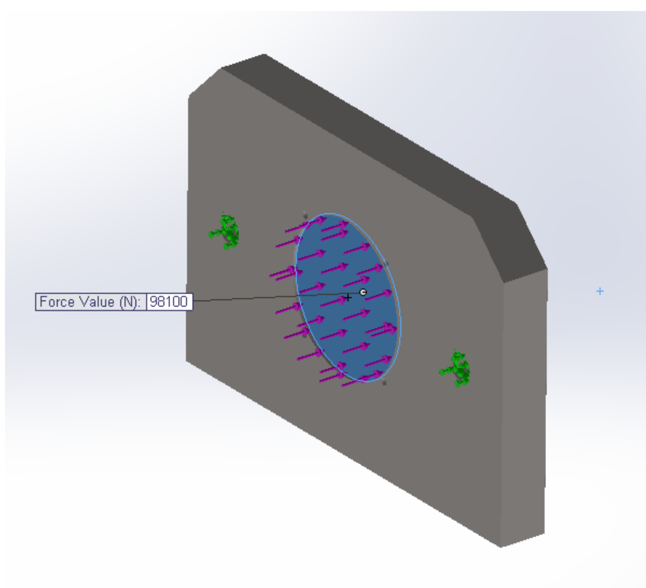


Figure 56 Boundary conditions –force 98100N applied and fixed at the face

6.19Stress Induced in Fixed Frame

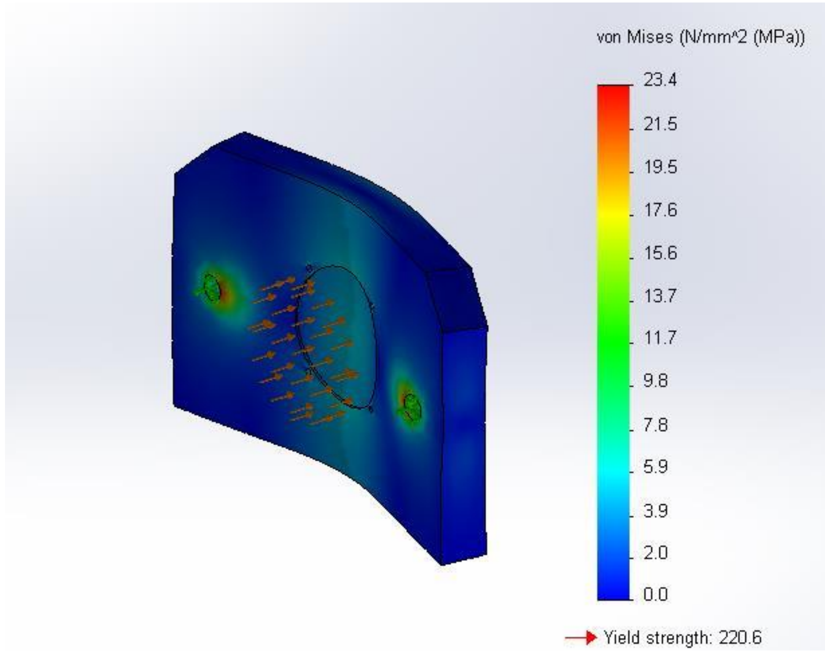


Figure 57 Stress 23.4 MPa induced in fixed frame

6.20 Deformation Induced in Fixed Frame

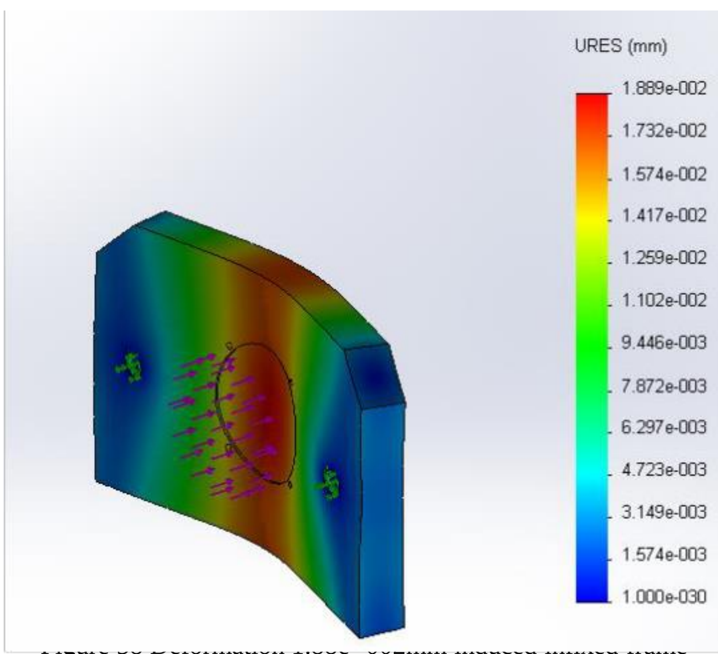


Figure 58 Deformation 1.889e-002mm induced infixed frame

6.21 3D Geometric Model View of Modified Moving Frame

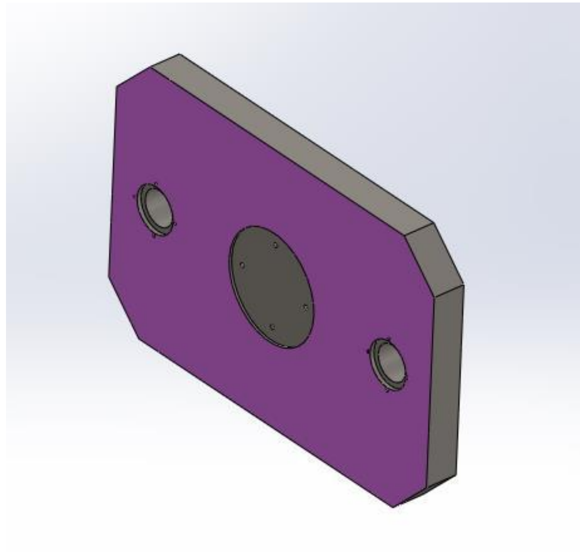


Figure 59 3D Geometric model view of modified moving frame

6.22 Meshed View of modified moving frame

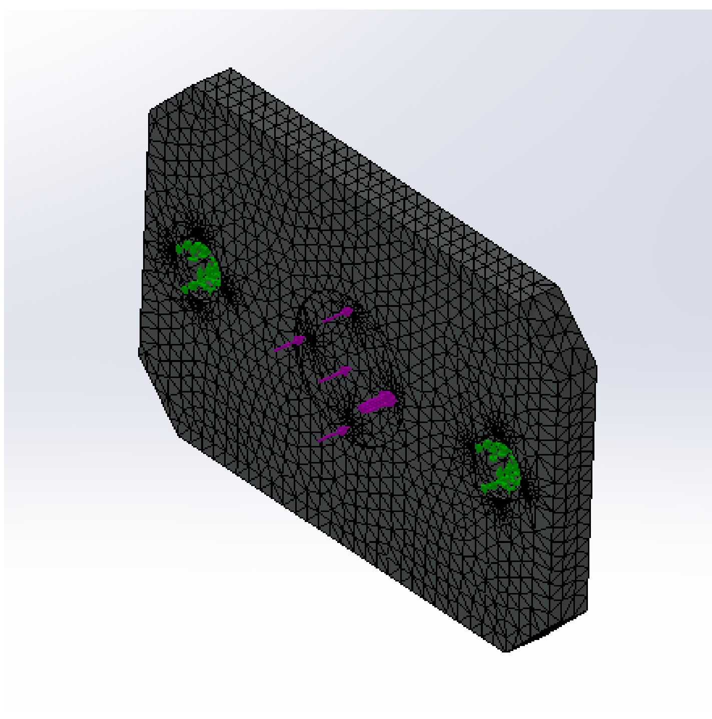


Figure 60 Meshed view of modified moving frame

6.23 Boundary Conditions– Force Applied and Fixed At the Face

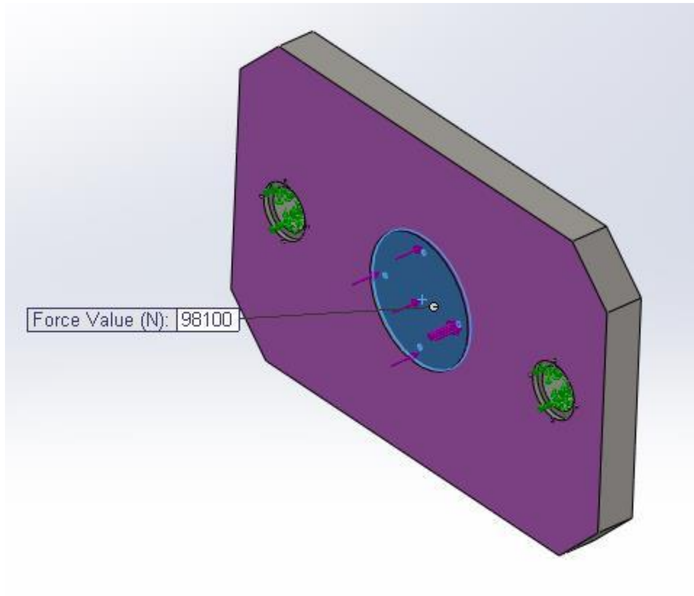
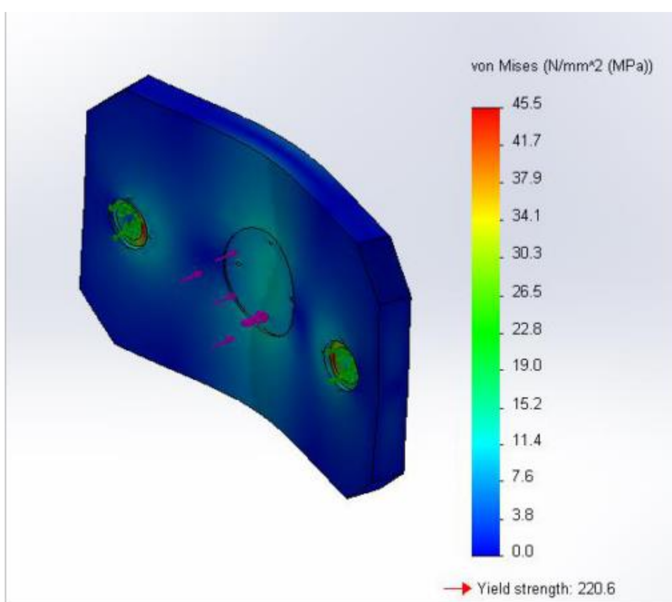


Figure 61 Boundary conditions –force 98100N applied and fixed at the face

6.24 Stress Induced in Modified Moving Frame



6.25 Deformation Induced in Modified Moving frame

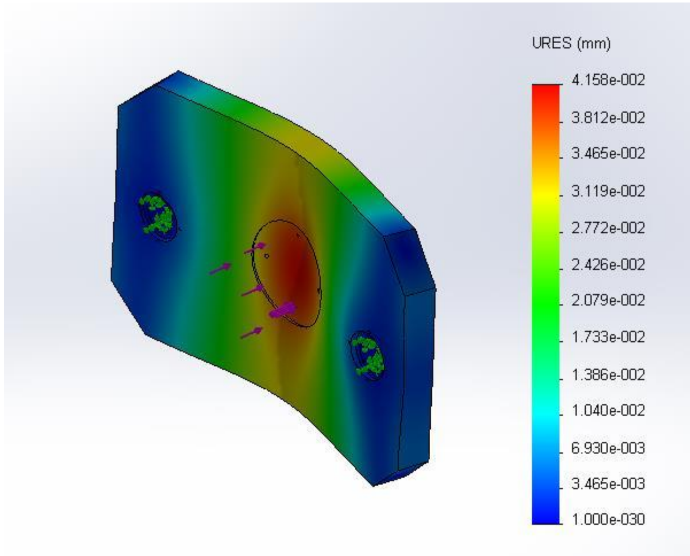


Figure 63 Deformation 4.158e -002 mm induced in modified moving frame

6.26 RESULT TABLE.

Table 1: shows result

Sl No	Description	Stress in N/mm2	Strain	FOS
1	Tie Rod	27.0	1.611e-001	8.14
2	Moving frame	178.7	3.571e-001	1.23
3	Fixed frame	23.4	1.889e-002	9.40
4	Cylinder	79.0	4.537e-002	2.78
5	Modified moving frame	45.5	4.158e-002	4.83

As per Design requirement we have maintained FOS above2.5 as this equipment is testing of valves no much deformation is recommended.

CHAPTER -7

CONCLUSION

An attempt was made to Design and Analysis the 10T Valve testingequipment and is Been done using Solid works software with simulation.

The 10T Valve Testing equipment is successfully designed to meet the requirements as per the constraints.

- With reference to the assumptions made above the maximum stress induced in the valve testing equipment is79 MPa it is less than the allowable stress & gives Factor of Safety is 2.78

- As this is valve testing equipment so higher factor of safety is recommended.
- Result fond by Analysis and Analytical ware closely resembles.

Scope for the further studies

Optimization is carried out to reduce weight and cost of the equipment without compromising the performance of the valve test equipment by using different material properties other Advanced Software. So that cost should not go higher than the present design.

REFERENCES

Books:

1. D. Dallas, 'Principles and Practices of Design and Construction' , Onward Press, First Edition,1994

2. R.S.Khurmi, A text book of Machine Design , India 3.

P.S.G. Design Data book, Coimbatore

4. "V.B.Bhandari", A book of "Design of Machine Elements", McGrawHill Publishing Co.Ltd, New Delhi, 2007

Websites:

5. www.solidworks.com.2.

6 http://www.valve-world.net/pdf/API_598_Valves_Inspection_and_Testing.pdf

7 https://azwardiir.files.wordpress.com/2010/06/apltcl025_sgd_l-01-hydraulic-fund