ISSN 2395-1621

A
Dissertation Report On
"Pressure Vessel Filter: FEA case Study"

Prof. D.K Nath

Savitri Bai Phule Pune University, Pune

ABSTRACT

Pressure vessels are designed to operate safely at "Design Pressure" .Cylindrical or spherical pressure vessels (e.g., hydraulic cylinders, gun barrels, pipes, boilers and tanks) are commonly used in industry to carry both liquids and gases under pressure.. Pressure vessel Filters are used in Chemical Processing plants ,Natural gas plants. Filtration is the separation of a fluid-solids solution involving the passing of this mixture through a porous barrier which retains most of the solid particulates contained in the mixture. The function of the filter sheet assembly is to prevent the leakage of natural gas under all circumstances since it is a highly inflammable gas.

The aim of this project is to design and analyse the pressure vessel filter assembly. In current filter the issues are coagulation in filter the internal pressure increases with time. Due to this internal pressure the tube sheet in filter is deform some time it will sheet in filter is deform some time it will break and the joint also break. This Analysis is used for Validation of This Analysis is used for Validation of FEA results with tested (actual) results.

Complete project runs through the following steps:

1. **Study of components of existing filter sheet assembly.**
2. **Study of pressures of various working conditions coming on filter sheet assembly.**
3. **Understanding the design of filter sheet. Finite element analysis of existing filter sheet and assembly.**
4. **From the nonlinear static structural analysis of existing filter sheet and assembly the area of maximum and minimum stress have been located.**
5. **Comparison of the basic designed model for conditions of filtering capacity, safety, cost.**

Keywords- pressure vessels, pressure vessel design, Pressure vessel filter types, Finite element analysis, Filter assembly, Ansys.

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I. INTRODUCTION

1.1 Introduction of Pressure Vessel

Pressure vessels are designed to operate safely at a specific pressure technically referred to as the "Design Pressure". A vessel that is inadequately designed to handle a high pressure constitutes a very significant safety hazard. Because of that, the design and certification of pressure vessels is governed by design codes such as the ASME Boiler and Pressure Vessel Code in North America, the Pressure Equipment Directive of the EU (PED), Japanese Industrial Standard (JIS), CSA B51 in Canada, Australian Standards in Australia and other international standards like Lloyd's, Germanischer Lloyd, Det Norske Veritas, Societe Generale de Surveillance (SGS S.A.), Stoomwezen etc

Cylindrical or spherical pressure vessels (e.g., hydraulic cylinders, gun barrels, pipes, boilers and tanks) are commonly used in industry to carry both liquids and gases under pressure. When the pressure vessel is exposed to this pressure, the material comprising the vessel is subjected to pressure loading, and hence stresses, from all directions.

Figure 1.1 : Pressure Vessel

The normal stresses resulting from this pressure are functions of the radius of the element under consideration, the shape of the pressure vessel (i.e., open ended cylinder, closed end cylinder, or sphere) as well as the applied pressure.

Theoretically almost any material with good tensile properties that is chemically stable in the chosen application could be employed. However, pressure vessel design codes and application standards (ASME BPVC Section II, EN 13445-2 etc.) contain long lists of approved materials with associated limitations in temperature range. Many pressure vessels are made of steel. To manufacture a cylindrical or spherical pressure vessel, rolled and possibly forged parts would have to be welded together. Some mechanical properties of steel, achieved by rolling or forging, could be adversely affected by welding, unless special precautions are taken. In addition to adequate mechanical strength, current standards dictate the use of steel with a high impact resistance, especially for vessels used in low temperatures. In applications where carbon steel would suffer corrosion, special corrosion resistant material should also be used. Some pressure vessels are made of composite materials, such as filament wound composite using carbon fibre held in place with a polymer. Due to the very high tensile strength of carbon fibre these vessels can be very light, but are much more difficult to manufacture. The composite material may be wound around a metal liner, forming a composite overwrapped pressure vessel. Other very common materials include polymers such as PET in carbonated beverage containers and copper in plumbing. Pressure vessels may be lined with various metals, ceramics, or polymers to prevent leaking and protect the structure of the vessel from the contained medium. This liner may also carry a significant portion of the pressure load.

The pressure vessels (i.e. cylinder or tanks) are used to store fluids under pressure. The fluid being stored may undergo a change of state inside the pressure vessel as in case of steam boilers or it may combine with other reagents as in a chemical plant. The pressure vessels are designed with great care because rupture of pressure vessels means an explosion which may cause loss of life and property. The material of pressure vessels may be brittle such that cast iron or ductile such as mild steel. Pressure vessels are used in a variety of applications in both industry and the private sector. They appear in these sectors as industrial compressed air receivers and domestic hot water storage tanks. Other examples of pressure vessels are diving cylinder, recompression chamber, distillation towers, autoclaves, and many other vessels in oil refineries and petrochemical plants, nuclear reactor vessel, pneumatic reservoir, hydraulic reservoir under pressure, road vehicle airbrake reservoir and storage vessels for liquefied gases such as ammonia, chlorine, propane, butane, and LPG.

1.2 Introduction of Pressure Vessel Filter

The natural gas coming out of ores is highly contaminated with sand particles. For further processing of natural gas it is necessary to have contamination free gas. So the air filters are used to clean the gas. Sand contamination can not only wreck the pumps but erode the pipeline so filters have to be installed to exclude the sand from the gas. The filtration of gas in industries is essential for enhancing longevity and performance of processing line components.

Pressure vessel Filters are used in Chemical Processing plants used Reaction Vessels. High pressure mix equipment is applied in reaction of petroleum and chemical industrial extensively, such as: solid-liquid, liquid-liquid, gas-liquid etc. It should be high requirements for mixers. Filtration is the separation of a fluid-solids solution involving the passing of this mixture through a porous barrier which retains most of the solid particulates contained in the mixture.

An air filter is a device used to decontaminate air that contains suspended impurities. A major application is in form of filtration of natural gases in petrochemical industries. Air filters are generally simple in concept yet they are technically complicated devices. The filter helps to mitigate the health effects of such gas contaminants by improving the quality of air or protects air handling equipment or costly systems like gas turbine compressors. The function of the filter sheet assembly is to prevent the leakage of natural gas under all circumstances since it is a highly inflammable gas. Besides, the environment around a refinery is extremely hazardous since the possibility of an explosion is always imminent. Thus, efficient functioning of filter sheet assembly extremely critical to a refinery. The refining industries require to design and analyze the filter assembly.

Fig 1.2 shows an existing filter unit used in oil refinery. Filter candles have been developed to provide optimal filtration performance combined with excellent flow pressure loss characteristics, mechanical strength and corrosion resistance. The candles can be configured as plain cylindrical or pleated elements in lengths up to 3m. Pulsed jet reverse cleaning can be carried out in-situ without interrupting forward flow and the filters have integral ventures to optimize performance. Filter candles can be supplied fully welded, screwed or flanged and bolted to a tube sheet. The connection provides a high integrity gasketed and threaded joint, enabling elements of filters to be easily assembled to the tube sheet. Support plates are fitted to the base of the tube sheet to provide lateral support and prevent element vibration. Tube sheet assemblies can be supplied to retrofit existing applications for relevant pressure vessel codes.

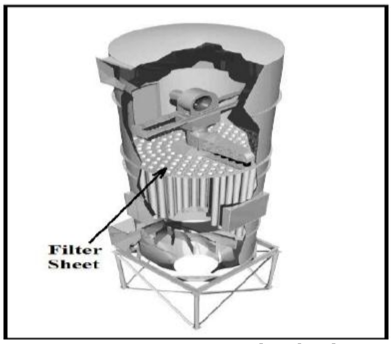


Figure 1.2 : Pressure Vessel with Filter

1.3 Background of the Project

The function of the filter sheet assembly is to prevent the leakage of natural gas under all circumstances since it is a highly inflammable gas. Besides, the environment around a refinery is extremely hazardous since the possibility of an explosion is always imminent. Thus, efficient functioning of filter sheet assembly is extremely critical to a refinery. The refining industries require to design and analyze the filter assembly for achieving following properties:

- High mechanical strength for longer filter life.
- Low pressure drop for less interruption of process flow.
- Greater corrosion resistance.
- Clean ability by pulse jet or back flushing to reduce maintenance costs.
- Increased filtration area, particularly with pleated elements.
- Increased dirt-holding capacity,
- Low in weight.

1.4 Current Problem

- Coagulation in filter the internal pressure increases with time.
- Due to this internal pressure the tube sheet in filter is deform some time it will sheet in filter is deform some time it will break and the joint also break.
- Maintenance is difficult in current design of filter sheet plate.
- To avoid this model of the tube sheet is require to improve.
- The Existing tube sheet Filter assembly require to analysis for finding stresses, deflection, strength require in joints etc.
- This Analysis is used for Validation of This Analysis is used for Validation of FEA results with tested (actual) results.
- Finite element based modelling is the best way to improve model of tube sheet assembly.

1.5 Scope of the Project

This project aims at design and analysis of the proposed model of the filter sheet assembly to find out stress and deflection in its various components using FEA. The complete project runs through the following steps:

- Study of components of existing filter sheet assembly.
- Study of pressures of various working conditions coming on filter sheet assembly.
- Understanding the design of filter sheet.
- Finite element analysis of existing filter sheet and assembly. From the nonlinear static structural analysis of existing filter sheet and assembly the area of maximum and minimum stress have been located.
- Comparison of the basic designed model for conditions of filtering capacity, safety, cost.

1.6 Objective

Current design has separate filters and mixing chambers, with the added cost, there is also additional risk in piping.

The objective of the project is to design, a single chamber, with a combined Filter and Mixer is proposed. 1/6th of the chamber has been already designed and tested; now the remaining portion is to be designed. So code shall be applied to all new design, which stipulate, that the structure should be safe at 2.5 times of operating pressure, which shall be tested using hydro test. A hydrostatic test is a way in which leaks can be found in pressure vessels. Hydro test is taking place on the actual pressure vessel. The 2.5 time operating pressure is passed through pressure vessel. The work can be done by using FEA (Finite element analysis) software. So the flows of the project work are as follow,

- To learn about basic concepts of FEA and design concept of thin pressure vessel.
- To learn ANSYS software.
- Model each section of Pressure Vessel Filter Sheet Plate using CAD software.
- Assemble the Each Part of the filter sheet Plate.
- Meshing, Material properties and Element type to be defined in ANSYS WORKBENCH.
- Apply boundary conditions.
- Perform structural analysis carried out in ANSYS WORKBENCH.
- Check the various values of von-mises stresses and deformation of filter sheet plate for allowable stress of 210Mpa at factor of safety 5.
- If values of von-mises stress is more than 42Mpa then repeat the procedure.

1.7 Methodology

In this dissertation work, it is proposed to carry out hydro test simulation of pressure vessel mixer cum filter. The position of the filter sheet plate is as shown in fig below

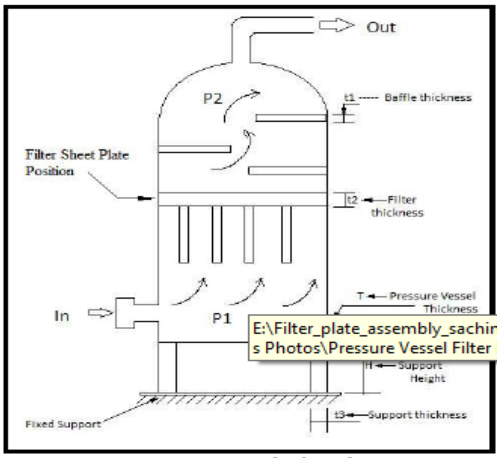


Figure 1.3 : Pressure Vessel Filter Plate Position

The dissertation work is planned in following phases.

Phase I:

1) Study the literature on pressure vessel design.
2) Study the literature on pressure vessel cum filter design.

Phase II:

1) Identify the critical parameters for designing the pressure vessel.

Phase III:

1) Prepare the CAD model using suitable modeling software
- For Pressure Vessel
- For Pressure Vessel with Filter
2) Analyze the model using FEA software for the pressure range 0.3 MPa to 0.5 MPa.
- Identifying critical locations where stresses are maximum.
- Validating the FEM results with available results in literatures for standard conditions.
- Filter is introduced in the pressure vessel & FEM analysis is carried out.

Phase IV:

1) Analysis is carried out by changing
- Geometry (Circular, Square cross section)
- Dimensions (Height & Diameter)
- Materials.
2) Determining optimum dimensions for a maximum applied pressure.

Phase V:

From the above work carried out in phase III and phase IV the graphs are plotted for various parameters & the conclusion is drawn.

II. Literature Review

Based on the literature review carried out, the contribution to the research pertaining to pressure vessel is as discussed below.

David Heckman et al [1]has shown the effect of using various elements like shell element, solid element, contact element under the assumption of model as complete solid, axi-symmetric and symmetric on run time & errors obtained in meridonial and circumferential stresses. He conclude that use of axi-symmetric elements produces lesser run time & error and obtained results from FEM where compared with hand calculations.

Wang Yang et al [2] studied that the failure of a stainless steel (SS) hydro-processing reactor that is used for flavor-fragrance test .The authors identifies the failure of hydro-processing reactor by various methods like macro-inspecting, Metallographic inspection for finding the cracks on the surface of parent material, cracks near to the circumferential weld.

Wang Yang et al [2] concludes that according to the discussion on the failure reason in the hydrogen processing reactor, the cracking in this reactor is a chloride induced stress corrosion cracking and the chloride ions were from the new catalyst. Also they recommend the following points to avoid the cracking in the hydrogen processing reactor, these are decreasing the concentration of chloride in the catalyst, selecting more corrosion resistant material, Reducing the welding residual stress & peening treatment on the inner surface of reactor.

J. E. Meyer [3] showed that the stress analysis of thin shell pressure vessel and the use of the simplified thin shell methods are illustrated by application to a pressure vessel that has many of the geometric and operational features of a pressurized water reactor vessel. He designed, constructed and operated so that a catastrophic failure is incredible. And find out the various stresses like pressure stresses, thermal stresses and discontinuity stresses on the various locations like long cylinder, sphere. B. Vivich et al [4] has demonstrated the correlation s between the linear FEA, non-linear FEA and actual test data. By applying proper contact conditions obtain the FEA result. By defining the different types of restraints such as immovable restraints, slider/roller restraints and displacement restraints applied to contact set up. After conducting such experiments on different types of modified contact set up, they conclude that stress concentration and normal forces are increases and show that FEA is very effective tool in designing and testing.

R.C. Carbonari et al [5] has discusses shape optimization of axisymmetric pressure vessels considering an integrated approach in which the entire pressure vessel model is used in conjunction with a multi-objective function that aims to minimize the von-Mises mechanical stress from nozzle to head. They conclude that For vessels with mechanical loading only and initial shapes close to a semi-sphere, one notices that the maximum head stresses are smaller than the cylinder stresses.

Z Sanal [6] showed that the nonlinear FEA requires careful engineering judgement, experience and powerful analysis software.Z Sanal represents the pressure vessel problems in his paper. Where the large displacement and plastic straining response of the structure is simulated by geometrically and materially nonlinear finite element analysis.

Shafique M. A. Khan [7] showed that the analysis results of stress distributions in a horizontal pressure vessel and the saddle supports. He obtained the results from 3-D Finite Element Analysis. Also he presents the stress distribution in the pressure vessel; the results provide details of stress distribution in different parts of the saddle separately, i.e. wear, web, flange and base plates. He showed that the effect of changing the load and various geometric parameters is investigated and recommendations are made for the optimal values of ratio of the distance of support from the end of the vessel to the length of the vessel and ratio of the length of the vessel to the radius of the vessel for minimum stresses both in the pressure vessel and the saddle structure.

Shafique M. A. Khan [7] conclude that the highly stressed area, beside the pressure vessel at the saddle horn, is the flange plate of the saddle. Yogesh Borse et al [11] deals with the Finite element modeling of Pressure vessels. Authors have tried to develop a finite element model taking due consideration on welding involved at the end connections of cylinder to shell end in modelling using shell elements to model cylinder. Theoretical investigation and engineering applications shows some junctions are rational in structure, convenient in fabrication and less in cost. Thus some end connections are tested under FEA for the cause of resulting weight reduction of 30-35 %. In this paper authors, describes its basic structure and the engineering finite element modelling for analyzing, testing and validation of pressure vessels under high stress zones.

Based on the literature review carried out, the contribution to the research pertaining to pressure vessel Filter sheet Plate is as discussed below.

V. G. Ukadgaonker et al [12] showed the different techniques for the analysis of filter sheet plate ie- tube sheet. These techniques are Analytical technique, Experimental technique and Numerical technique. In Analytical technique he and Ligament Efficiency which takes into account the weakening effect of the perforations. In Experimental technique he conducted experiment on orated in a square pattern and were tested for uniaxial loading in both the pitch and diagonal directions. Plates with four different ligament efficiencies ranging between 13% and 50% were tested. And in Numerical technique Jones has determined the elastic stress distribution in the perforated plate with triangular penetration pattern for in-plane loads and bending loads. 3D analysis was made for a plate with 5% ligament efficiency and 2D analysis for a plate with 10% ligament efficiency.

W. J. O'Donnell et al [13] describes the method for calculating stresses and deflections in perforated plates with a triangular penetration pattern. The method is based partly on theory and partly on experiment. Average ligament stresses are obtained from purely theoretical considerations but effective elastic constants and peak stresses are derived from strain measurements and photo elastic tests. Acceptable limits for pressure stresses and thermal stresses in heat-exchanger tube sheets are also proposed

III. FINITE ELEMENT ANALYSIS

3.1 Introduction

The finite element method (FEM), sometimes referred to as *finite element analysis* (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. Boundary value problems are also sometimes called *field* problems. The field is the domain of interest and most often represents a physical structure. The *field variables* are the dependent variables of interest governed by the differential equation. The *boundary conditions* are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field. Depending on the type of physical problem being analyzed, the field variables may include physical displacement, temperature, heat flux, and fluid velocity to name only a few.

3.2 A General Procedure for FEA

Certain steps in formulating a finite element analysis of a physical problem are common to all such analyses, whether structural, heat transfer, fluid flow, or some other problem. These steps are embodied in commercial finite element software packages (some are mentioned in the following paragraphs) and are implicitly incorporated in this text, although we do not necessarily refer to the steps explicitly in the following chapters. The steps are described as follows.

3.2.1 Preprocessing

The preprocessing step is, quite generally, described as defining the model and includes

- Define the geometric domain of the problem.
- Define the element type(s) to be used.
- Define the material properties of the elements.
- Define the geometric properties of the elements (length, area, and the like).
- Define the element connectivity's (mesh the model).
- Define the physical constraints (boundary conditions).
- Define the loadings.

The preprocessing (model definition) step is critical. In no case is there a better example of the computer-related axiom "garbage in, garbage out." A perfectly computed finite element solution is of absolutely no value if it corresponds to the wrong problem.

3.2.2 Solution

During the solution phase, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary field variable(s). The computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow.

As it is not uncommon for a finite element model to be represented by tens of thousands of equations, special solution techniques are used to reduce data storage requirements and computation time. For static, linear problems, a wave front solver, based on Gauss elimination is commonly used.

3.2.3 Post-processing

Analysis and evaluation of the solution results is referred to as post-processing. Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution. Examples of operations that can be accomplished include

- Sort element stresses in order of magnitude.
- Check equilibrium.
- Calculate factors of safety.
- Plot deformed structural shape.
- Animate dynamic model behavior.
- Produce color-coded temperature plots.

While solution data can be manipulated many ways in post processing, the most important objective is to apply sound engineering judgment in determining whether the solution results are physically reasonable.

3.3 Brief History of Finite Element Method

The mathematical roots of the finite element method dates back at least a half century. Approximate methods for solving differential equations using trial solutions are even older in origin. Lord Rayleigh and Ritz used trial functions (in our context, interpolation functions) to approximate solutions of differential equations. Galerkin used the same concept for solution. The drawback in the earlier approaches, compared to the modern finite element method, is that the trial functions must apply over the entire domain of the problem of concern.

While the Galerkin method provides a very strong basis for the finite element method, not until the 1940s, when Courant introduced the concept of piecewise-continuous functions in a sub domain, did the finite element method have its real start.

In the late 1940s, aircraft engineers were dealing with the invention of the jet engine and the needs for more sophisticated analysis of airframe structures to withstand larger loads associated with higher speeds. These engineers, without the benefit of modern computers, developed matrix methods of force analysis, collectively known as the flexibility method, in which the unknowns are the forces and the knowns are displacements. The finite element method, in its most often-used form, corresponds to the displacement method, in which the unknowns are system displacements in response to applied force systems. In this text, we adhere exclusively to the displacement method. As will be seen as we proceed, the term displacement is quite general in the finite element method and can represent physical displacement, temperature, or fluid velocity, for example

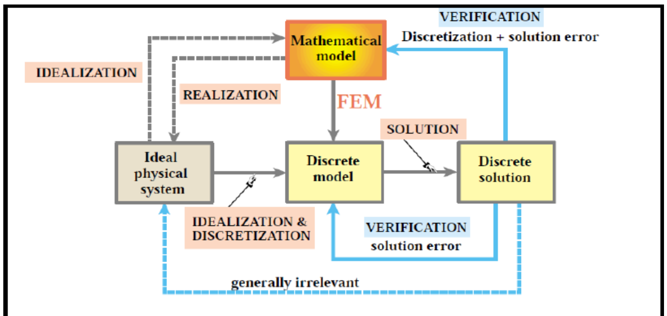
The term finite element was first used by Clough in 1960 in the context of plane stress analysis and has been in common usage since that time. During the decades of the 1960s and 1970s, the finite element method was extended to applications in plate bending, shell bending, pressure vessels, and general three-dimensional problems in elastic structural analysis as well as to fluid flow and heat transfer. Further extension of the method to large deflections and dynamic analysis also occurred during this time period. An excellent history of the finite element method and detailed bibliography is given by Noor.

The finite element method is computationally intensive, owing to the required operations on very large matrices. In the early years, applications were performed using mainframe computers, which, at the time, were considered to be very powerful, high-speed tools for use in engineering analysis. During the 1960s, the finite element software code NASTRAN was developed in conjunction with the space exploration program of the United States.

NASTRAN was the first major finite element software code. It was, and still is, capable of hundreds of thousands of degrees of freedom (nodal field variable computations). In the years since the development of NASTRAN, many commercial software packages have been introduced for finite element analysis. Among these are ANSYS, ALGOR, and COSMOS/M. In today's computational environment, most of these packages can be used on desktop computers and engineering workstations to obtain solutions to large problems in static and dynamic structural analysis, heat transfer, fluid flow, electromagnetic, and seismic response. In this text, we do not utilize or champion a particular code. Rather, we develop the fundamentals for understanding of finite element analysis to enable the reader to use such software packages with an educated understanding.

3.4 The FEM Analysis Process

A model-based simulation process using FEM involves doing a sequence of steps. This sequence takes two canonical configurations depending on the environment in which FEM is used. These are reviewed next to introduce terminology



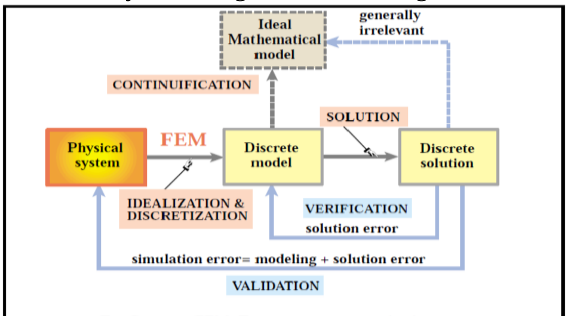
The Mathematical FEM. The mathematical model (top) is the source of the simulation process. Discrete model and solution follow from it. The ideal physical system (should one go to the trouble of exhibiting it) is inessential.

Figure 3.1:FEM Analysis Process

3.4.1 The Mathematical FEM

The process steps are illustrated in Figure 3.2. The process centerpiece, from which everything emanates, is the mathematical model. This is often an ordinary or partial differential equation in space and time. A discrete finite element model is generated from a variation or weak form of the mathematical model. This is the discretization step. The FEM equations are processed by an equation solver, which delivers a discrete solution (or solutions).On the left Figure 3.2 shows an ideal physical system. This may be presented as a realization of the mathematical model; conversely, the mathematical model is said to be an idealization of this system. For example, if the mathematical model is the Poisson's equation, realizations may be heat conduction or a electrostatic charge distribution problem. This step is inessential and may be left out. Indeed FEM discretizations may be constructed without any reference to physics.

The concept of error arises when the discrete solution is substituted in the "model" boxes. This replacement is generically called verification. The solution error is the amount by which the discrete solution fails to satisfy the discrete equations. This error is relatively unimportant when using computers, and in particular direct linear equation solvers, for the solution step. More relevant is the discretization error, which is the amount by which the discrete solution fails to satisfy the mathematical model. Replacing into the ideal physical system would in principle quantify modeling errors. In the mathematical FEM this is largely irrelevant, however, because the ideal physical system is merely that: a figment of the imagination.



The Physical FEM. The physical system (left) is the source of the simulation process. The ideal mathematical model (should one go to the trouble of constructing it) is inessential.

Figure 3.2: Mathematical FEM

3.4.2 The Physical FEM

The second way of using FEM is the process illustrated in Figure 3.3. The centerpiece is now the physical system to be modeled. Accordingly, this sequence is called the Physical FEM. The processes of idealization and discretization are carried out concurrently to produce the discrete model. The solution is computed as before. Just like Figure 3.2 shows an ideal physical system, 4.3 depicts an ideal mathematical model. This may be presented as a continuum limit or “continuification” of the discrete model. For some physical systems, notably those well modeled by continuum fields, this step is useful. For others, such as complex engineering systems, it makes no sense. Indeed FEM discretizations may be constructed and adjusted without reference to mathematical models, simply from experimental measurements. The concept of error arises in the Physical FEM in two ways, known as verification and validation, respectively.Verification is the same as in the Mathematical FEM: the discrete solution is replaced into the discrete model to get the solution error. As noted above this error is not generally important.

Substitution in the ideal mathematical model in principle provides the discretization error. This is rarely useful in complex engineering systems, however, because there is no reason to expect that the mathematical model exists, and if it does, that it is more physically relevant than the discrete model.

Validation tries to compare the discrete solution against observation by computing the simulation error, which combines modeling and solution errors. As the latter is typically insignificant, the simulation error in practice can be identified with the modeling error.

One-way to adjust the discrete model so that it represents the physics better is called model updating.

The discrete model is given free parameters. These are determined by comparing the discrete solution against experiments, as illustrated in Figure 3.3. Inasmuch as the minimization conditions are generally nonlinear (even if the model is linear) the updating process is inherently iterative.

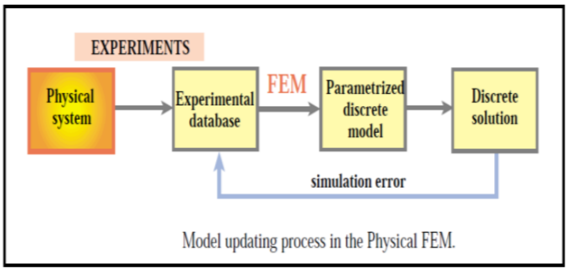


Figure 3.3: Physical FEM

3.4.3 Synergy of Physical and Mathematical FEM

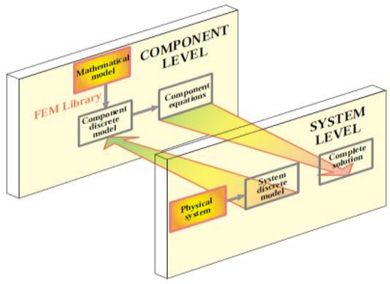
The foregoing physical and mathematical sequences are not exclusive but complementary. This synergy is one of the reasons behind the power and acceptance of the method. Historically the Physical FEM was the first one to be developed to model very complex systems such as aircraft. The Mathematical FEM came later and, among other things, provided the necessary theoretical underpinnings to extend FEM beyond structural analysis.

A glance at the schematics of a commercial jet aircraft makes obvious the reasons behind the physical FEM. There is no differential equation that captures, at a continuum mechanics level, the structure, avionics, fuel, propulsion, cargo, and passengers eating dinner.

There is no reason for despair, however. The time honored divides and conquer strategy, coupled with abstraction, comes to the rescue. First, separate the structure and view the rest as masses and forces, most of which are time-varying and nondeterministic. Second, consider the aircraft structure as built of substructures: 10 wings, fuselage, stabilizers, engines, landing gears, and so on.

Take each substructure, and continue to decompose it into components: rings, ribs, spars, cover plates, actuators, etc, continuing through as many levels as necessary. Eventually those components become sufficiently simple in geometry and connectivity that they can be reasonably well described by the continuum mathematical models provided, for instance, by Mechanics of Materials or the Theory of Elasticity. At that point, stop. The component level discrete equations are obtained from a FEM library based on the mathematical model. The system model is obtained by going through the reverse process: from component equations to substructure equations, and from those to the equations of the complete aircraft. This system assembly process is governed by the classical principles of Newtonian mechanics expressed in conservation form.

This multilevel decomposition process is diagrammed in Figure 3.4, in which the intermediate substructure level is omitted for simplicity.



Combining physical and mathematical modeling through multilevel FEM. Only two levels (system and component) are shown for simplicity; intermediate substructure levels are omitted.

Figure 3.4: Synergy of Physical and Mathematical FEM

3.5 FEA Elements

There are different elements used in FEA. Properties of these elements are given as follows:

3.5.1 Intrinsic Dimensionality

Elements can have one, two or three space dimensions. There are also special elements with zero dimensionality, such as lumped springs or point masses.

3.6 Nodal points

Each element possesses a set of distinguishing points called nodal points or nodes for short. Nodes serve two purposes: definition of element geometry, and home for degrees of freedom. They are usually located at the corners or end points of elements, as illustrated in Figure 3.5; in the so-called refined or higher-order elements nodes are also placed on sides or faces, as well as the interior of the element

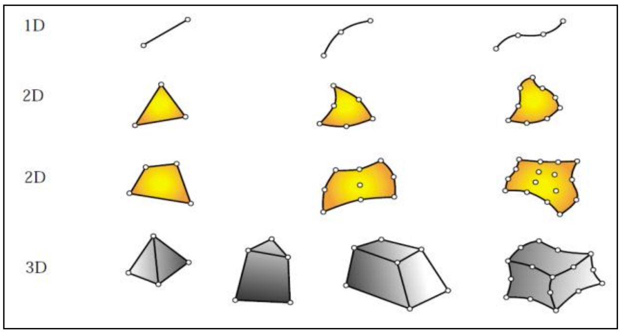


Figure 3.5: Typical finite element geometries in one through three dimensions

3.6.1 Geometry

The geometry of the element is defined by the placement of the nodal points. Most elements used in practice have fairly simple geometries. In one-dimension, elements are usually straight lines or curved segments. In two dimensions they are of triangular or quadrilateral shape.

In three dimensions the three common shapes are tetrahedral, pentahedral (also called wedges or prisms), and hexahedra (also called cuboids or "bricks"). See Figure 3.5

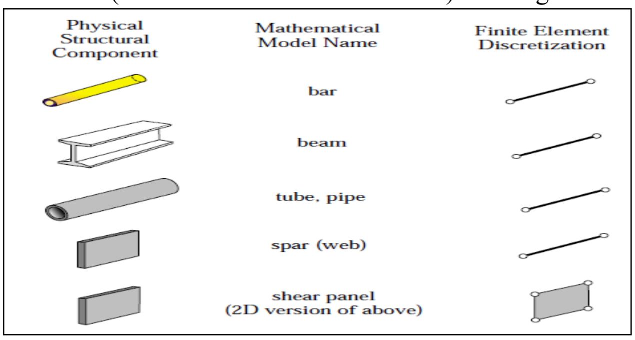


Figure 3.6 : Examples of primitive structural elements

3.6.2 Degrees of freedom

The degrees of freedom (DOF) specify the state of the element. They also function as "handles" through which adjacent elements are connected. DOFs are defined as the values (and possibly derivatives) of a primary field variable at nodal points. The actual selection depends on criteria studied at length in Part II. Here we simply note that the key factor is the way in which the primary variable appears in the mathematical model. For mechanical elements, the primary variable is the displacement field and the DOF for many (but not all) elements are the displacement components at the nodes.

3.6.3 Nodal forces

There is always a set of nodal forces in a one-to-one correspondence with degrees of freedom. In mechanical elements the correspondence is established through energy arguments.

3.6.4 Constitutive properties

For a mechanical element these are relations that specify the material behavior. For example, in a linear elastic bar element it is sufficient to specify the elastic modulus E and the thermal coefficient of expansion α.

3.6.5 Fabrication properties

For mechanical elements these are fabrication properties which have been integrated out from the element dimensionality. Examples are cross sectional properties of MoM elements such as bars, beams and shafts, as well as the thickness of a plate or shell element

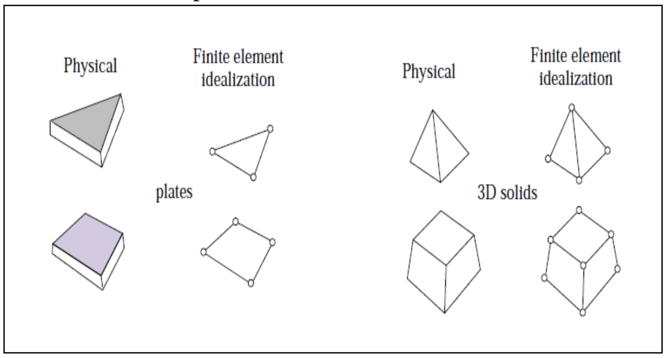


Figure 3.7 : Continuum element examples

3.7 Classification of Mechanical Elements

The following classification of finite elements in structural mechanics is loosely based on the "closeness" of the element with respect to the original physical structure. It is given here because it clarifies many points that appear over and over in subsequent sections and provides insight into advanced modeling techniques such as hierarchical breakdown and global-local analysis.

3.7.1 Primitive Structural Elements

These resemble fabricated structural components. The qualifier primitive is used to distinguish them from macro-elements, which is another element class described below. It means that they are not decomposable into simpler elements. These elements are usually derived from Mechanics-of-Materials simplified theories and are better understood from a physical, rather than mathematical, standpoint.

3.7.2 Continuum Elements

These do not resemble fabricated structural components at all. They result from the subdivision of "blobs" of continua, or of structural components viewed as continua. Unlike structural elements, continuum elements are better understood in terms of their mathematical interpretation. Examples: plates, slices, shells, axi-symmetric solids, general solids.

3.7.3 Special Elements

Special elements partake of the characteristics of structural and continuum elements. They are derived from a continuum mechanics standpoint but include features closely related to the physics of the problem. Examples: crack elements for fracture mechanics applications, shear panels, infinite and semi-infinite elements, contact and penalty elements, rigid-body elements. See Figure 3.8.

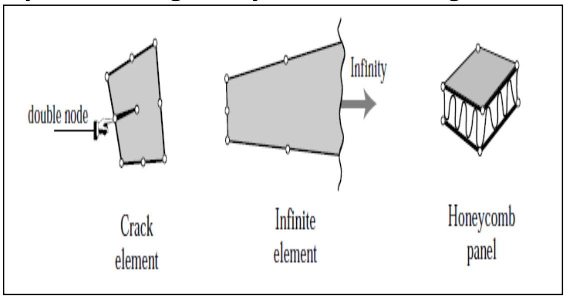


Figure 3.8: Special element examples

3.7.4 Macro elements

Macro-elements are also called mesh units and super-elements, although the latter term overlaps with substructures (defined below). These often resemble structural components, but are fabricated with simpler elements.

3.7.5 Substructures

These are also called structural modules and super elements. These are macro elements with a well defined structural function, typically obtained by cutting the complete structure into functional components. Examples: the wings and fuselage in an airplane, the deck and cables in a suspension bridge. It should be noted that the distinction between complete structures, substructures and macro elements is not clear-cut. The term super element is often used in a collective sense to embrace all levels beyond that of primitive elements.

3.8 Nonlinear Structural Analysis

Structural nonlinearities occur on a routine basis. For example, whenever you staple two pieces of paper together, the metal staples are permanently bent into a different shape, as shown in Figure 3.9: Common Examples of Nonlinear Structural Behavior

(a) If you heavily load a wooden shelf, it will sag more and more as time passes,

(b) As weight is added to a car or truck, the contact surfaces between its pneumatic tires and the underlying pavement change in response to the added load,

(c) If you were to plot the load-deflection curve for each example, you would discover that they exhibit the fundamental characteristic of nonlinear structural behaviour a changing structural stiffness.

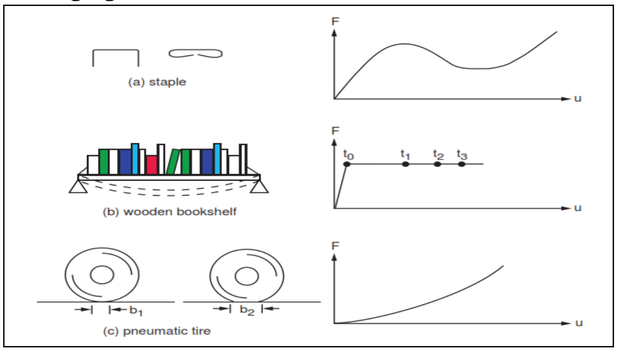


Figure 3.9: Common Examples of Nonlinear Structural Behavior

The following nonlinear structural analysis topics are available:

3.8.1 Causes of Nonlinear Behavior

Nonlinear structural behavior arises from a number of causes, which can be grouped into these principal categories:

- Changing status
- Geometric nonlinearities
- Material nonlinearities

3.8.2 Changing Status (Including Contact)

Many common structural features exhibit nonlinear behavior that is status-dependent. For example, a tension only cable is either slack or taut; a roller support is either in contact or not in contact. Status changes might be directly related to load (as in the case of the cable), or they might be determined by some external cause. Situations in which contact occurs are common to many different nonlinear applications. Contact forms a distinctive and important subset to the category of changing-status nonlinearities.See the Contact Technology Guide for detailed information on performing contact analyses using ANSYS.

3.8.3 Geometric Nonlinearities

If a structure experiences large deformations, its changing geometric configuration can cause the structure to respond nonlinearly. An example would be the fishing rod shown in Figure 3.10: A Fishing Rod Demonstrates Geometric Nonlinearity. Geometric nonlinearity is characterized by "large" displacements and/or rotations.

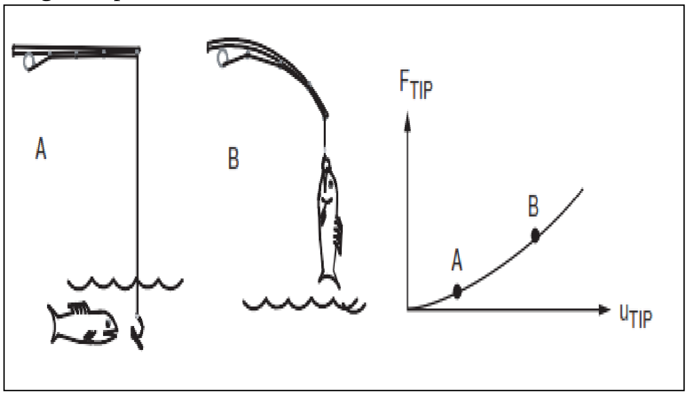


Figure 3.10 : A Fishing Rod Demonstrates Geometric Nonlinearity

3.8.4 Material Nonlinearities

Nonlinear stress-strain relationships are a common cause of nonlinear structural behavior. Many factors can influence a material's stress-strain properties, including load history (as in elastoplastic response), environmental conditions (such as temperature), and the amount of time that a load is applied (as in creep response).

3.8.5 Basic Information about Nonlinear Analyses

ANSYS employs the "Newton-Raphson" approach to solve nonlinear problems. In this approach, the load is subdivided into a series of load increments. The load increments can be applied over several load steps. Figure 3.11: Newton-Raphson Approach illustrates the use of Newton-Raphson equilibrium iterations in a single DOF nonlinear analysis.

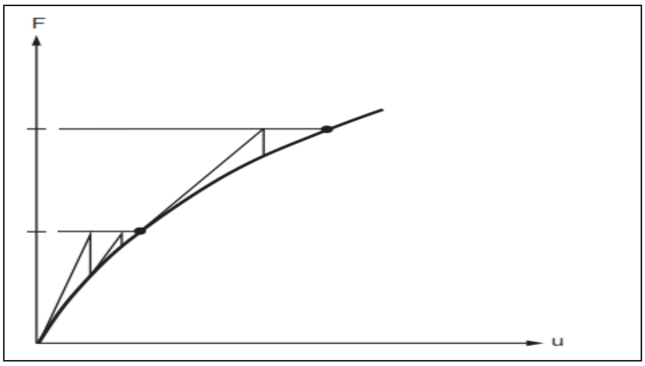


Figure 3.11: Newton-Raphson Approach

Before each solution, the Newton-Raphson method evaluates the out-of-balance load vector, which is the difference between the restoring forces (the loads corresponding to the element stresses) and the applied loads. The program then performs a linear solution, using the out-of-balance loads, and checks for convergence. If convergence criteria are not satisfied, the out-of-balance load vector is reevaluated, the stiffness matrix is updated, and a new solution is obtained. This iterative procedure continues until the problem converges.

A number of convergence-enhancement and recovery features, such as line search, automatic load stepping, and bisection, can be activated to help the problem to converge. If convergence cannot be achieved, then the program attempts to solve with a smaller load increment.

In some nonlinear static analyses, if you use the Newton-Raphson method alone, the tangent stiffness matrix may become singular (or non-unique), causing severe convergence difficulties. Such occurrences include nonlinear buckling analyses in which the structure either collapses completely or "snaps through" to another stable configuration. For such situations, you can activate an alternative iteration scheme, the arc-length method, to help avoid bifurcation points and track unloading.

The arc-length method causes the Newton-Raphson equilibrium iterations to converge along an arc, thereby often preventing divergence, even when the slope of the load vs. deflection curve becomes zero or negative. This iteration method is represented schematically in Figure 3.12: Traditional Newton-Raphson Method vs. Arc Length Method.

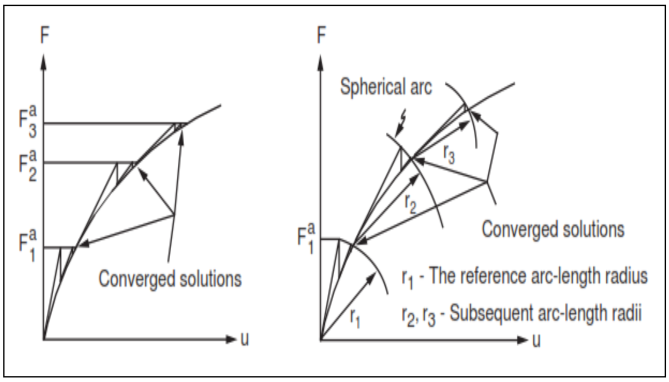


Figure 3.12: Traditional Newton-Raphson Vs. Arc-Length Method

To summarize, a nonlinear analysis is organized into three levels of operation:

• The "top" level consists of the load steps that you define explicitly over a "time" span (see the discussion of "time" in "Loading" in the Basic Analysis Guide). Loads are assumed to vary linearly within load steps (for static analyses).

• Within each load step, you can direct the program to perform several solutions (sub steps or time steps) to apply the load gradually.

• At each sub step, the program will perform a number of equilibrium iterations to obtain a converged solution

3.8.6 Overview of Static Structural Analysis

Static structural analysis is one in which the load/field conditions does not vary with time and the assumption here is that the load or field conditions are gradually applied (not suddenly applied). The most common application of FEA is the solution of stress related design problems. The behavior of the system could be either linear or non-linear.

Typically in structural analysis the kind of matrices solved is:

[K] * [X] = [F] ………………………… [3.1]

Where K is called the stiffness matrix, X is called the displacement matrix and F is the load matrix. This is a force balance equation. At times, the elements of matrix [K] are the function of [X]. Such system is called non-linear system. From a formal point of view, three conditions have to be met in any stress analysis, equilibrium of forces (or stresses), compatibility of displacements and satisfaction of the state of stress at continuum boundaries.

The kind of loads that can experience here could be:

1. Force load applied at one or several points
2. Pressure loads that can be distributed over one or multiple region
3. Inertia loads due to motion as a result of velocity, acceleration or deceleration
4. Thermal loads due to heating effects

The outputs that can be expected are:

1. Specified displacements applied at one or more locations
2. Displacement at one or more points
3. Strains at one or more points
4. Stresses at one or more points
5. Reaction forces

It all starts off with the formulation of the components 'stiffness' matrix. This square matrix is formed from details of the material properties, the model geometry and any assumptions of the stress-strain field (plane stress or strain). Once the stiffness matrix is created, it may be used with the knowledge of the forces to evaluate the displacement of the structure (hence the term displacement analysis). On evaluation of the displacement, they are differentiated to give six strains distributions, 3 mutually perpendicular direct strains and 3 corresponding shear strains.

3.8.7 Steps in Static Structural Analysis

There are basic steps in stress analysis for the accurate checking of the component. These steps are as follows:

- Definition of material Properties :
  The material properties can be inserted from engineering data files.
- Meshing :
  The meshing of the particular component is done from the MESH option from toolbar. The

refinement in meshing is done with RELAVANCE TOOL.

- Choose analysis type :
  The analysis type is chosen from NEW ANALYSIS toolbar, for stress analysis STATIC STRUCTURAL is chosen. Then define various input and output controls. From Data management settings specific solutions files can be saved from this analysis for further use. Also initial conditions are defined after this step.
- Apply Loads and Supports :
  The application of load is done from the types of various load types such as pressure, force, fixed support and displacement.
- Solution :
  The solution of the analysis is done from the SOLVE command.
- Review Results:
  All structural result types except frequencies are available from the results of a static structural analysis. By making use of a Solution Information object we can track, monitor and diagnose problems that arise during a solution. Once a solution is available we can outline the results or animate the results to review the of the structure. As a result of a nonlinear static analysis we can have a solution at several time periods.

IV. MODELLING OF FILTER SHEET PLATE ASSEMBLY

4.1 2D Drawing Of Filter Sheet Plate Assembly

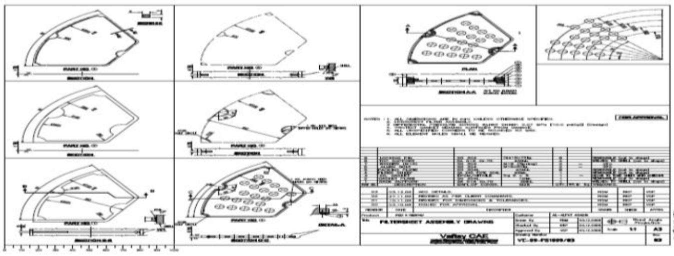


Figure 4.1 : Drawing of Filter Sheet Plate Assembly

Fig shows the complete 2-D drawings of the Filter sheet plate assembly and Bill of Material of the Filter sheet Plate Assembly.

4.2 3-D Model of Filter Sheet Plate Assembly

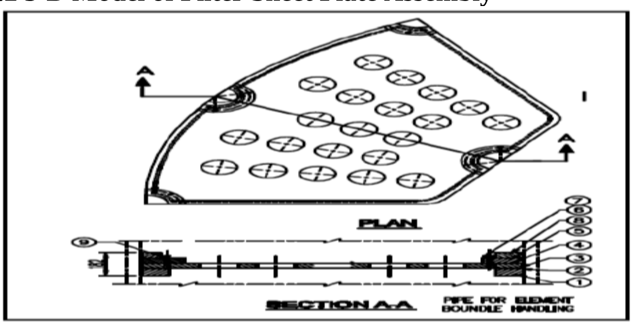


Figure 4.2 : Detail Drawing of Filter Sheet Plate Assembly

Modeling of Filter Sheet Plate and its assembly is carried out by using CAD software in CATIA V5 R18.
In the current existing Model, the BASE SUPPORT & TOP SUPPORT are welded to shell (cut to shape). In this assembly GASKET PLATE is inserted above the BASE SUPPORT. SQUARE GASKET made up of EPDM/NITRILE material is glude with adhesive to the FILTER SHEET. Then FILTER SHEET is inserted between the TOP PLATE & BASE SUPPORT. FILTER SHEET removable from the assemble for the purpose of any maintenance. The PACKING PLATE is inserted between the FILTER SHEET & TOP SUPPORT. The PACKING PLATE is also removable.

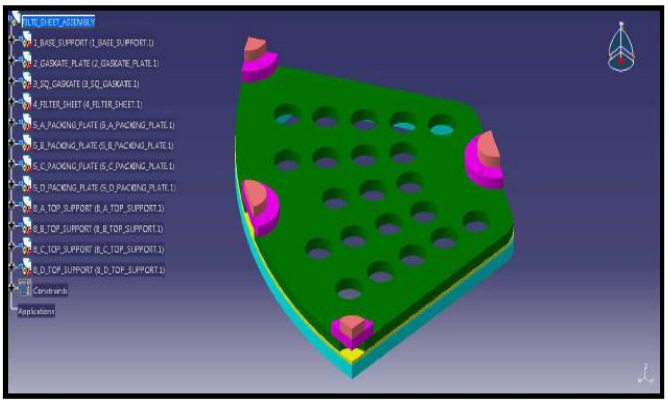


Figure 4.3 : 3-D Model of Filter Sheet Plate Assembly

4.2.1Bill of Material

PART NO.	DESCRIPTION	MATL OF CONST.	SIZE	QTY	REMARK
1	BASE SUPPORT	SA 516 Gr. 70	50 thk	1	WELDED TO SHELL (cut to shape)
2	GASKET PLATE	SS 304L	12 thk	1	REMOVABLE (cut to shape)
3	SQ. GASKET	EPDM / NITRILE	Sq. 8 x 8	1	GLUDE TO TUBE SHEET WITH ADHESIVE
4	FILTER SHEET	SA 240 TYPE 304L	40 thk	1	REMOVABLE (cut to shape)
5	PACKING PLATE	SS 304L	40 thk	6	REMOVABLE (cut to shape)
6	ALLEN BOLT	SS 304	M16 x 60lg	2	STD
7	WASHER (M16)	SS 304	M16 (spring)	2	STD
8	TOP SUPPORT	SA 516 Gr. 70	40 thk	4	WELDED TO SHELL (cut to shape)
9	LOCKING PIN	SS 304	Dia 16 x 75 lg	6	REMOVABLE (cut to shape)

Table 4-1 : Bill Of Material

4.2.2 Filter Sheet Plate Assembly Parts

Modeling of Filter Sheet Plate parts are as shown in fig below.

4.2.2.1 Base Support

Base support is welded to pressure vessel shell.

Quantity: 1

Size: 50 mm thick

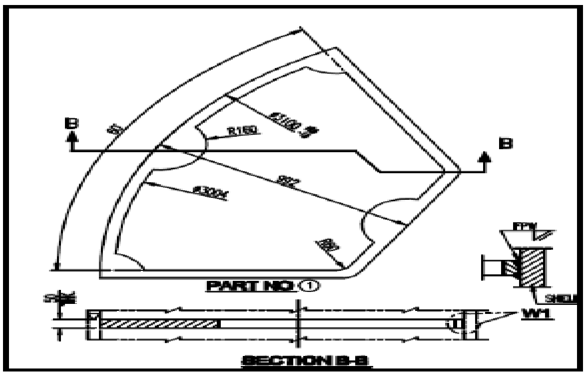


Figure 4.4 : Drawing of Base Support

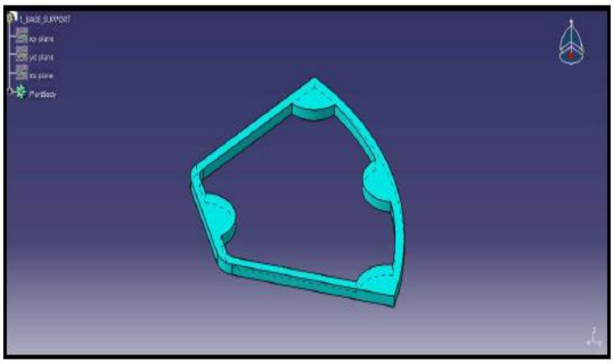


Figure 4.5 : 3-D Model of Base Support

4.2.2.2 Gasket Plate

Gasket plate is removable from assembly of pressure vessel filter sheet plate assembly. It is not welded.

Quantity: 1

Size: 50 mm thick

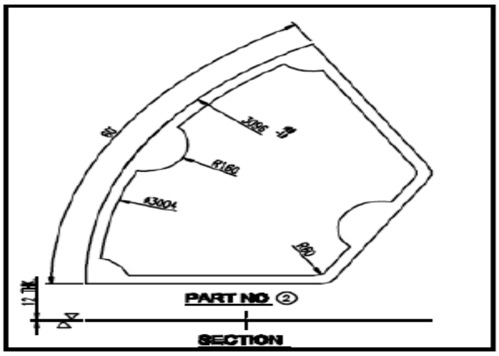


Figure 4.6: Drawing of Gasket Plate

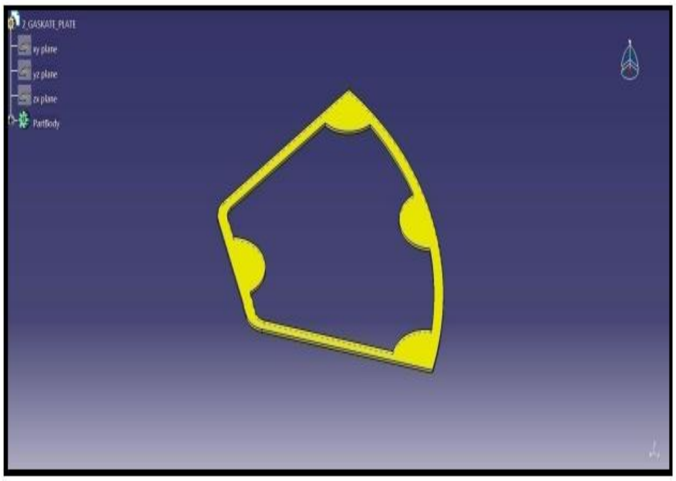


Figure 4.7: 3-D Model of Gasket Plate

4.2.2.3 Square Gasket

Square Gasket is glude to tube sheet with adhesive.

Quantity: 1

Size: Square (8 x 8) mm

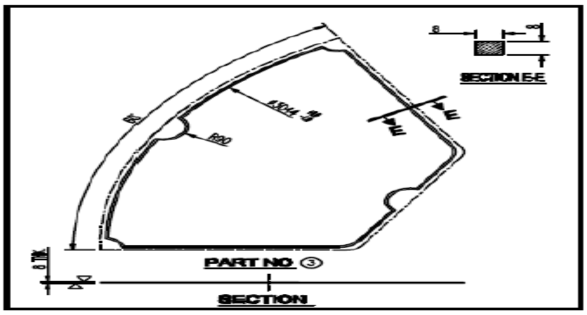


Figure 4.8: Drawing of Square Gasket

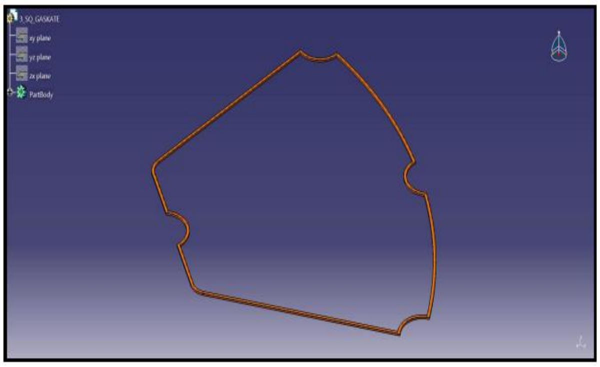


Figure 4.9 : 3-D Model of Square Gasket

4.2.2.4 Filter Sheet

Filter Sheet is removable from assembly of pressure vessel filter sheet plate assembly. It is not welded.

Quantity: 1

Size: 40 mm thick

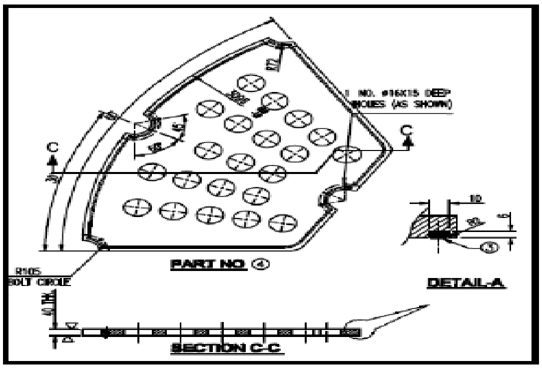


Figure 4.10 : Drawing of Filter Sheet

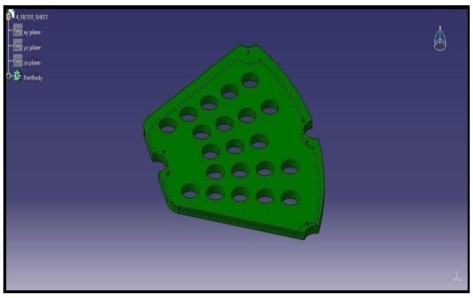


Figure 4.11: 3-D Model of Filter Sheet

4.2.2.5 Packing Plate

Packing plate is removable from assembly of pressure vessel filter sheet plate assembly. It is not welded.

Quantity: 6
Size: 40 mm thick

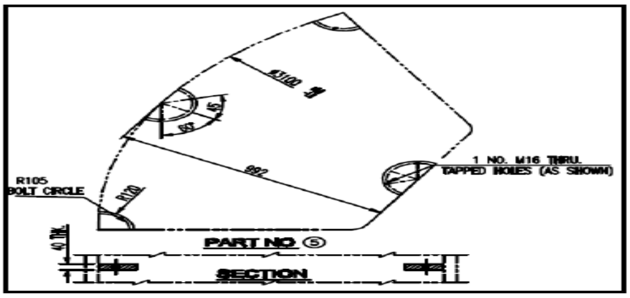


Figure 4.12: Drawing of Packing Plate

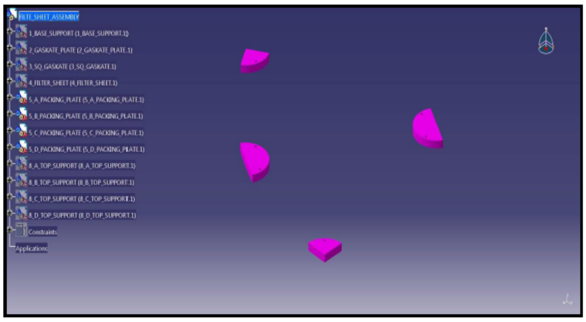


Figure 4.13 : 3-D Model of Packing Plates

4.2.2.6 Allen Bolt

Allen Bolt is a standard part and it was taken from Catlog standards.

Quantity: 2

Size: M16 x 16 mm long

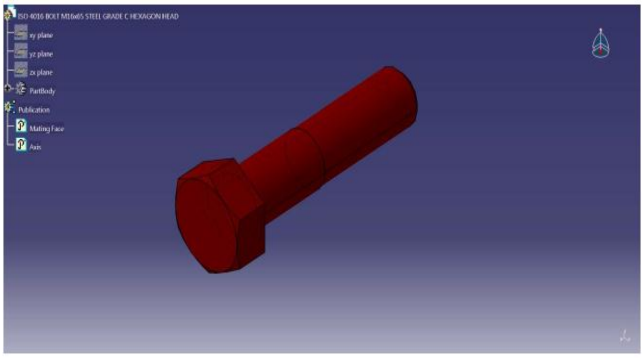


Figure 4.14 : 3-D Model of Allen Bolt

4.2.2.7 Washer (M16)

Washer is a standard part and it was taken from Catlog standards.

Quantity: 2
Size: M16 (spring)

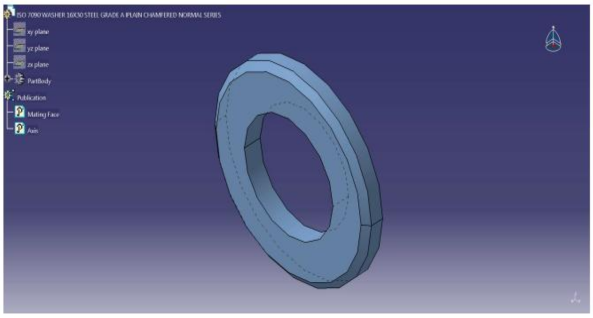


Figure 4.15 : 3-D Drawing Washer (M16)

4.2.2.8 Top Supports

Top support is welded to pressure vessel shell.

Quantity: 4

Size : 40 mm thick

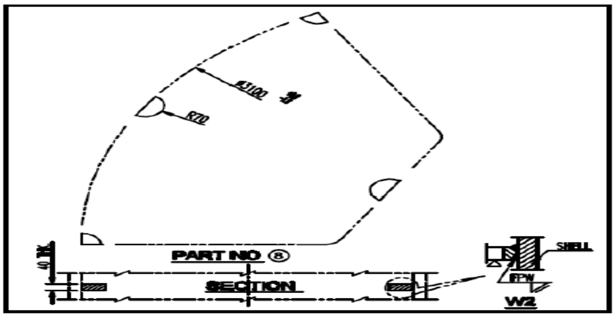


Figure 4.16 : Drawing of Top Support

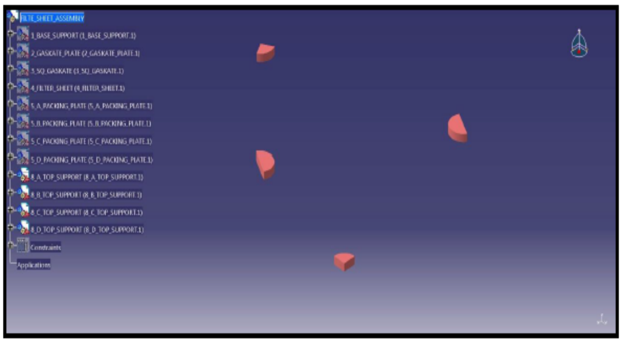


Figure 4.17 : 3-D Drawing of Top Support

4.2.2.9 Locking Pin

Locking Pin is a standard part and it was taken from Catlog standards.

Quantity: 6

Size : Dia 16 mm x 75 mm long

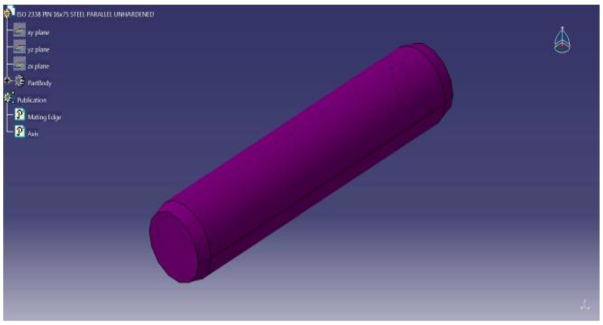


Figure 4.18 : 3-D Drawing of Locking Pin

V. ANALYSIS OF FILTER SHEET PLATE ASSEMBLY

For Analysis of the Filter Sheet Assembly here we consider the 1/6th part of the Full Filter Sheet Plate. Because the Filter sheet plate is symmetric. So Here we consider the 1/6th part of Filter Sheet Plate for ANALYSIS.

5.1 Need of Analysis

In industry the component produced may be of different sizes, from flat plates of very simple shape to complex 3 dimensional solid bodies. During operation they may be subjected to various types of applied loading conditions which include centrifugal force, pressure and temperature loading and prescribed boundary conditions. With rising cost of material over design, the resultant wastage may be extremely costly. Failure of component during service may produce a high service return rate, which is extremely undesirable both from high replacement cost and damage to the prestige of product. So stress analysis at the design stage is essential, if service failures are to be avoided and near optimum designs are to be achieved for specified operating conditions.

5.2 Steps in Analysis

Following are the steps in ANALYSIS for the filter sheet plate assembly.

- Analysis Type : Static Structural
- Engineering Data : Selection of material and material properties
- Model : Import the model from CATIA
- Meshing : Discritization of Filter Sheet Plate Assembly
- Boundary Condition : Apply boundary condition (Degree of freedom and Force)
- Solve: Solving for Getting Von mises stresses and Deformation.

5.3 Engineering Data

Following are the properties of the material:

- Material : Structural Steel
- Mass (*m*) : 424.68Kg
- Density (ρ) : 7850 Kg/m3
- Volu1me (*v*) : 5.433E7m3
- Young's Modulus (E) : 2E11 Pa
- Poisson's ratio (μ) : 0.3
- Yield strength : 2.5E8 Pa
- Factor of Safety (F.S.) : 5

5.4 Importing Geometry from CATIA

Model of existing Filter sheet plate assembly from CATIA is imported in ANSYS as shown in fig below.

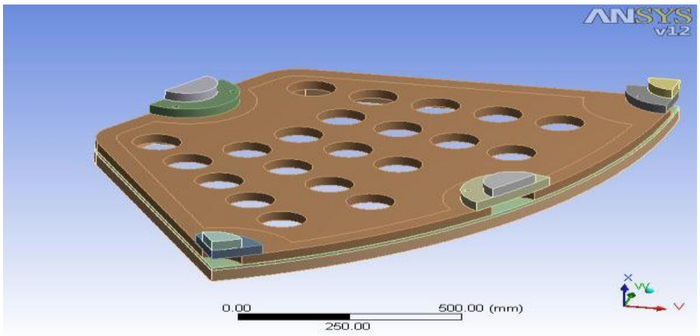


Figure 5.1: Imported Model of Filter sheet plate from CATIA

5.4.1 Steps for Importing Geometry from CATIA

Following are the steps for importing the model of Filter Sheet Plate Assembly Model from the CATIA of CAD modelling software to the ANSYS WORKBENCH CAE software.

- Create the CAD Model of Filter Sheet Plate Assembly in CAD software CATIA V5 R18.
- Save that CAD Model in *.stp format.
- Open the ANSYS WORKBENCH of CAE software.
- Import that CAD model of *.stp format file from the file menu of ANSYS WORBENCH software.

5.5 Meshing: Discretization of Filter Sheet Plate Assembly

Basic methodology of finite element method is to prepare calculations at only limited number of points and then interpolate the results for entire domain i.e. surface and volume. Any continuous object has infinite number of degrees of freedom and it is impossible to solve it for required results in this format. Finite element method reduces degrees of freedom from infinite to finite with the help of discretization of entire domain. Various types of elements like 1D, 2D, 3D, mass, spring, damper, gap etc. are available for meshing, one has to select them depending upon the geometry, size and shape of the component, type of the analysis to be carried out and time availability for completion of project.

5.5.1 Steps Involved in Meshing of Filter Sheet Plate Assembly in ANSYS

- Selection of element type for meshing
- Creating simplified parts
- Meshing the part
- Obtaining desirable mesh pattern for different size and quality.

5.5.1.1 Selection of Element Type for Meshing

The filter sheet plate assembly model is meshed with 10 node Tetrahedron SOLID187 element. SOLID187 has a quadratic displacement behavior and is well suited to modeling irregular meshes. The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyper elasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elasto plastic materials, and fully incompressible hyper elastic materials.

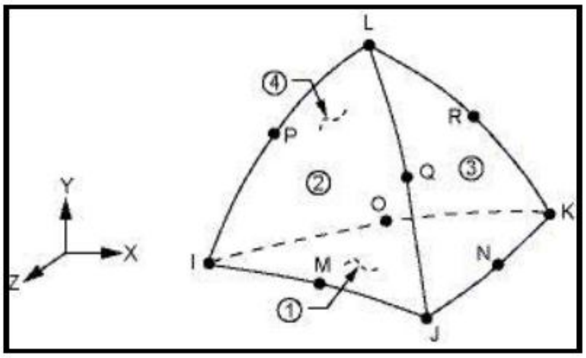


Figure 5.2 : SOLID 187 Geometry

5.5.1.2 Mesh Sensitivity Analysis

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, we have to remember 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.

5.5.1.3 Creating Simplified and Meshing the Parts

Element Type: SOLID 187

Method of Mesh Control:

a. Tetra Hedron

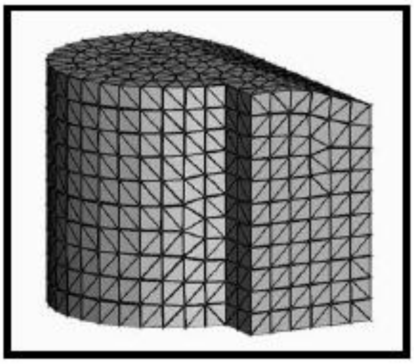


Figure 5.3 : Tetrahedron Mesh Sample

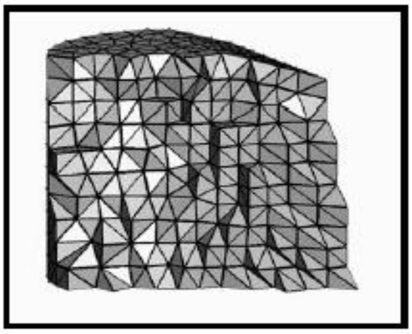


Figure 5.4 Tetrahedron Mesh Sample

Cross Section

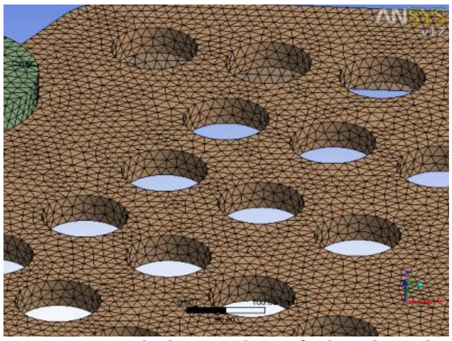


Figure 5.5 : Tetrahedron Meshing of Filter Sheet Plate

b. Hex-dominant

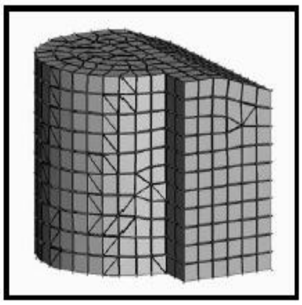


Figure 5.6: Hex Dominant Mesh Sample

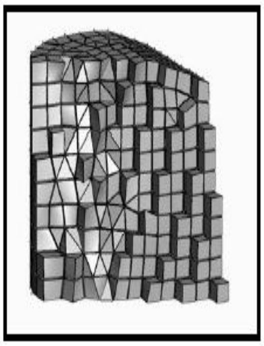


Figure 5.7 : Hex Dominant Mesh Sample

Cross Section

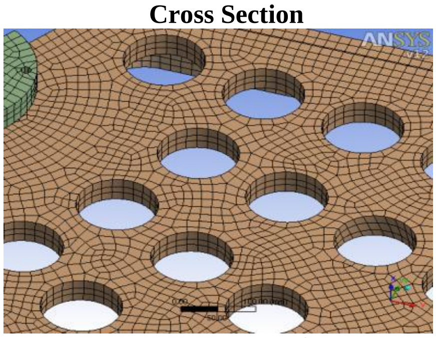


Figure 5.8 : Hex Dominant Meshing of Filter Sheet Plate

5.6 Different Patterns Used For Analysis

The Analysis of Filter Sheet plate can be done by changing the pattern of the filter tube hole by 30˚, 45˚, 60˚ & 90˚ patterns. Figure 5.8 shows different patterns of the filter tube holes.

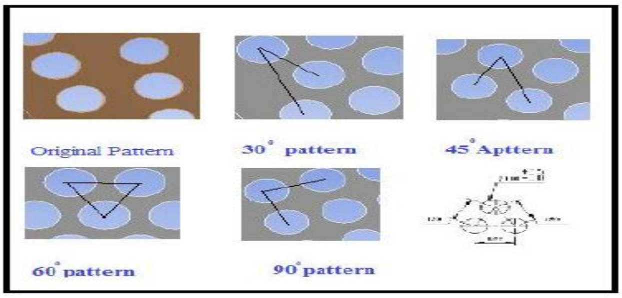


Figure 5.9: Different Pitch Patterns

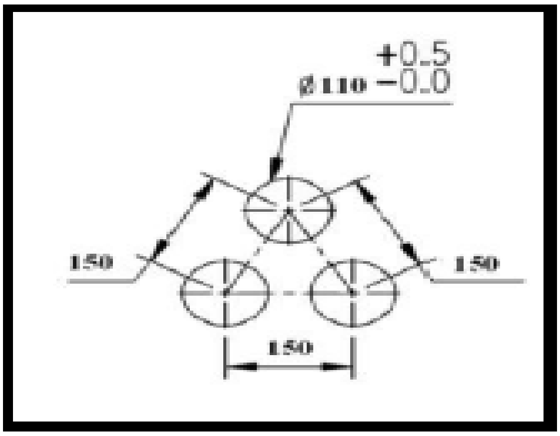


Figure 5.10: 2-D Drawing of Triangular pitch Pattern

Figure 5.10 shows the detail dimension of the triangular pitch pattern. And Figure 5.11 shows the Radial Pitch pattern of the Filter Sheet Plate.

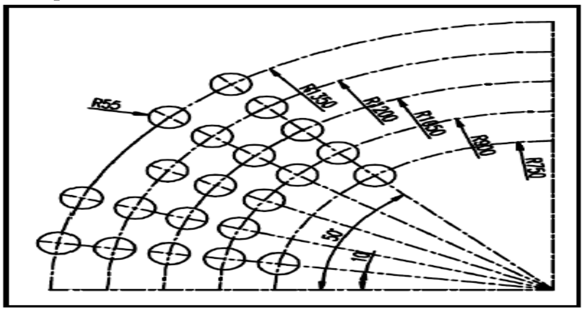


Figure 5.11: Radial Pitch Pattern of existing Filter Sheet Plate

Following are the different type of Patterns used for further Analysis Filter Sheet Plate.

1. Existing Filter tube hole Pattern on Filter sheet plate.

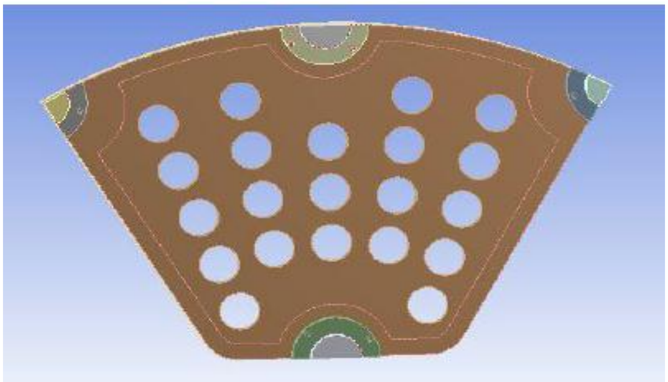


Figure 5.12 :Existing Filter Tube hole Pattern

2. 30˚ Filter tube hole pattern on Filter sheet plate.

Figure 5.13 :30˚ Filter Tube Hole Pattern

3. 45˚ Filter tube hole pattern on Filter sheet plate.

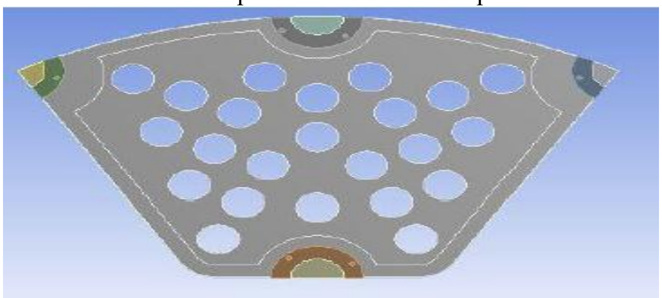


Figure5.14 : 45˚ Filter Tube Hole Pattern

4. 60˚ Filter tube hole pattern on Filter sheet plate.

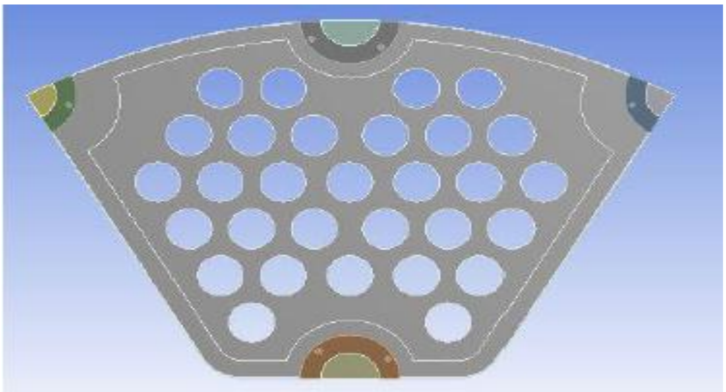


Figure5.15 : 60˚ Filter Tube Hole Pattern

5. 90˚ Filter tube hole pattern on Filter sheet plate.

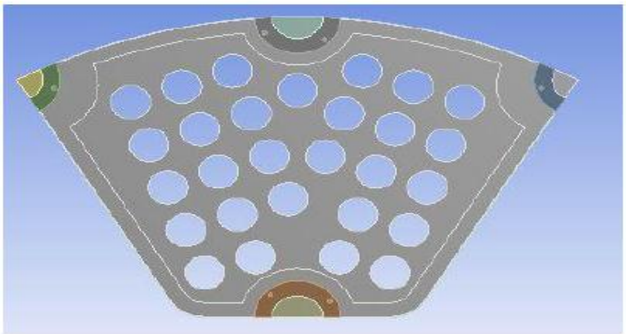


Figure5.16 : 90˚ Filter Tube Hole Pattern

5.7 Working Conditions

Working Pressure : 0.07 MPa

Hydrostatic Test Pressure : 2.5 * working Pressure
: (2.5 * 0.07) MPa
: 0.175 Mpa

Back Pressure : 0.08 MPa

5.8 Boundary Conditions

[1] Outer face of Top and bottom support is considered as Fixed (Welded rigidly to the outer shell).

[2] Shell boundary of chamber restricts the movement of filter sheet, gasket plate and packing plate in XY plane, hence we need to accommodate those conditions by giving displacements in the X and Y directions as zero).

5.8.1 Boundary Conditions for Different Cases of Analysis

Following are the different conditions for Different cases. The Details of Different cases is explained in Chapter 6

5.8.1.1 Boundary Condition for Case A

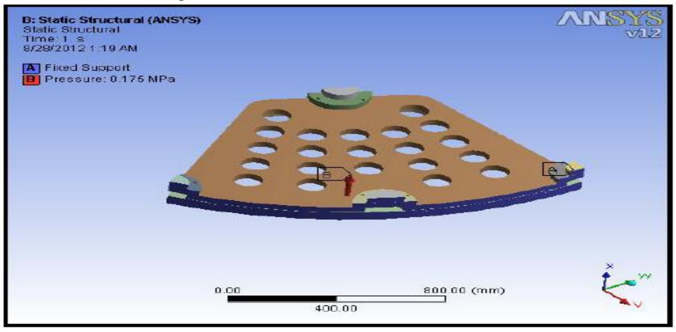


Figure 5.17 : Boundary Condition of Filter Sheet Plate Assembly in CASE A

5.8.1.2 Boundary Condition for Case B

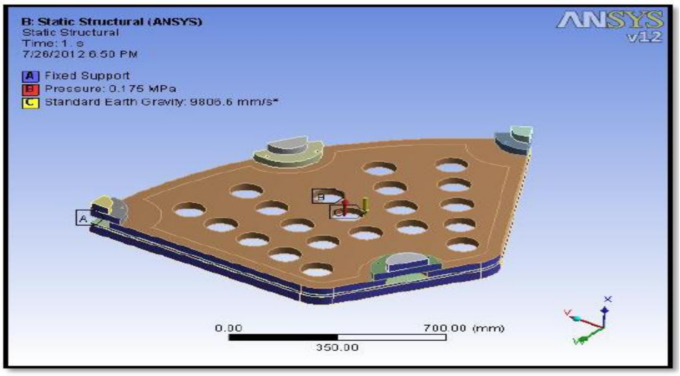


Figure 5.18: Boundary Condition of Filter Sheet Plate Assembly in CASE B

5.8.1.3 Boundary Condition for Case C

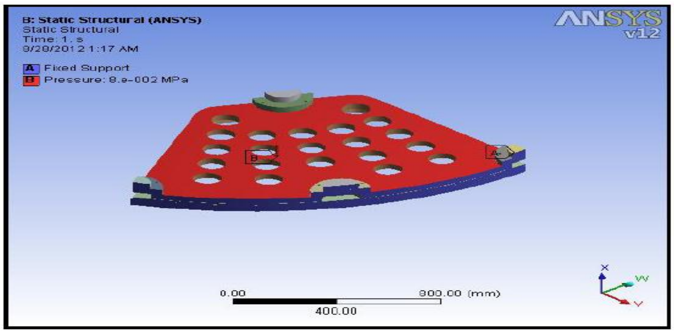


Figure 5.19: Boundary Condition of Filter Sheet Plate Assembly in CASE C

5.8.1.4 Boundary Condition for Case D

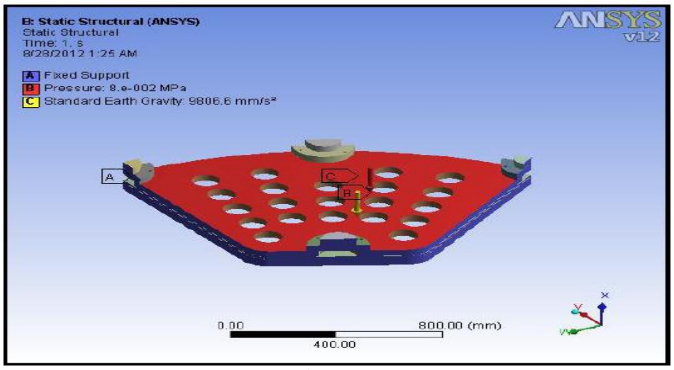


Figure 5.20: Boundary Condition of Filter Sheet Plate Assembly in CASE D

5.9 Solve

After giving the boundary conditions the model is solve for

a) Maximum Equivalent stress (Von mises stress)

b) Maximum Total deformation

VI. DIFFERENT CASES FOR ANALYSIS

6.1.1 Case A (Considering Hydrostatic Test Pressure only)

In Case A Non- Linear static Analysis is done and Hydrostatic Test Pressure of 0.175 Mpa is applied on Filter Sheet Plate Assembly. And got the different values of Von Mises Stresses & Deformations for Different values of Element size. And two different meshing control methods are used to meshing the Filter Sheet Plate Assembly viz. -

Tetrahedron meshing method & Hex- Dominant meshing method.

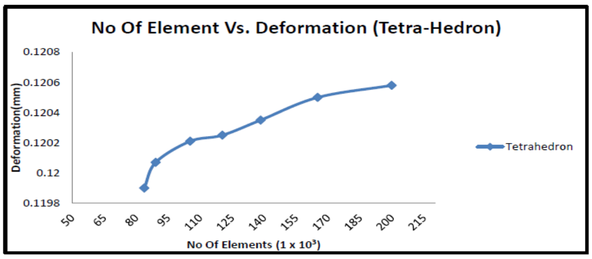
Filter Plate Pattern Used: Original Pattern as it is.

Sub steps: 5

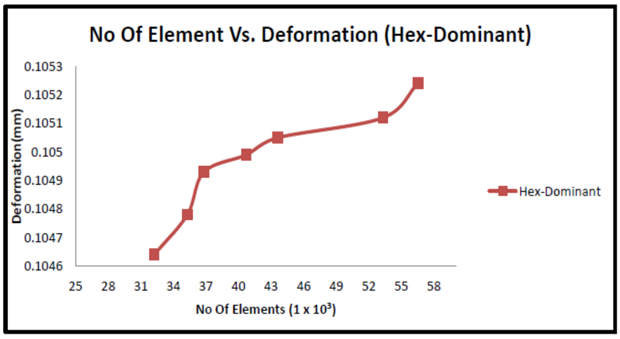
Meshing Type	Element Size(mm)	No of Element (1×10^3)	Deformation (mm)	Von-Mises Stress(Mpa)	Time (Sec)
Tetrahedron	20	83.712	0.1199	30.363	1709
	19	89.077	0.12007	30.125	2184
	18	105.244	0.12021	31.472	3871
	17	120.35	0.12025	31.619	4637
	16	138.313	0.12035	32.898	5710
	15	**165.034**	**0.1205**	**32.431**	**6243**
	14	199.736	0.12058	33.132	7150
Hex-Dominant	19	32.217	0.10464	30.124	1504
	18	35.267	0.10478	30.302	1690
	17	36.807	0.10493	32.857	1762
	16	40.694	0.10499	32.854	1871
	15	**43.606**	**0.10505**	**32.796**	**1967**
	14	53.296	0.10512	33.911	1980
	13	56.508	0.10524	33.189	2444

Table 6-1: Result Table for Case A

Graph fro Case A [no element VS Deformation]:



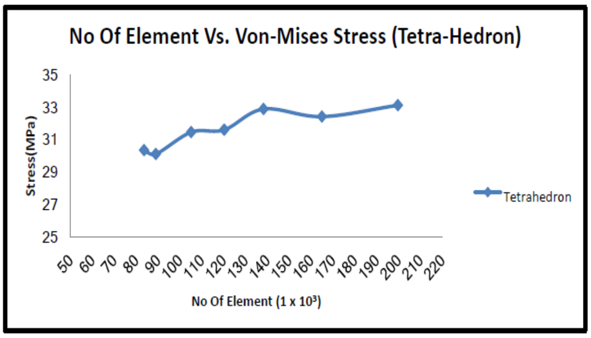
Graph 6-1: Graph For Case A of Tetra Hedron (No of Element Vs. Deformation)



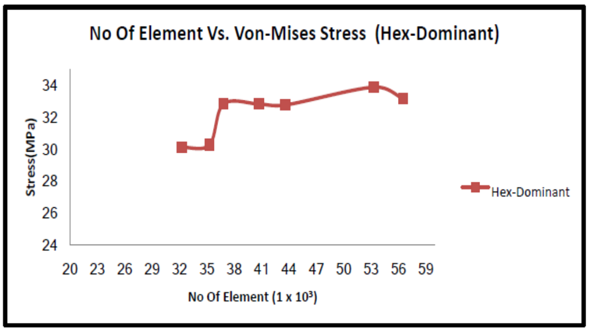
Graph 6-2: Graph For Case A of Hex Dominant (No of Element Vs.Deformation)

From the graph of No of element vs. deformation of the filter plate assembly in case A, it is clear that, the graph is converged after the element size 15(no of element for tetrahedron = 165034 & no of element for Hex dominant = 43606). So that the deformation value for the Non-Linear Static Analysis using tetrahedron method meshing is 0.12050 mm at element size 15 mm and deformation value by using the Hex-dominant method meshing is 0.10505 mm at element size 15 mm.

Graph For Case A [no of Elements VS von-mises stress]:



Graph 6-3: Graph for Case A of Tetra Hedron (No of Element Vs. Von Mises Stress)



Graph 6-4: Graph For Case A of Hex Dominant (No of Element Vs. Von Mises Stress)

From the graph of No of element vs. Von-Mises stress of the filter plate assembly in case A, it is clear that, the graph is converged after the element size 15(no of element for tetrahedron = 165034 & no of element for Hex dominant = 43606). So that the Von-Mises value for the Non-Linear Static Analysis using tetrahedron method meshing is 32.431 MPa at element size 15 mm and Von-Mises value by using the Hex-dominant method meshing is 32.796 MPa at element size 15 mm.

6.1.2 Analysis Snapshots for Case A (Considering Hydrostatic Test Pressure only)

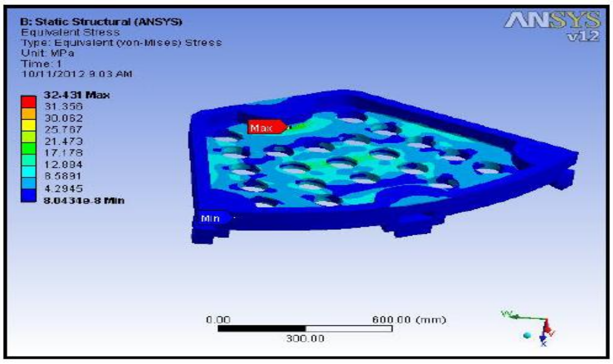


Figure 6.1: Maximum Stress for Case A (Tetra Hedron)

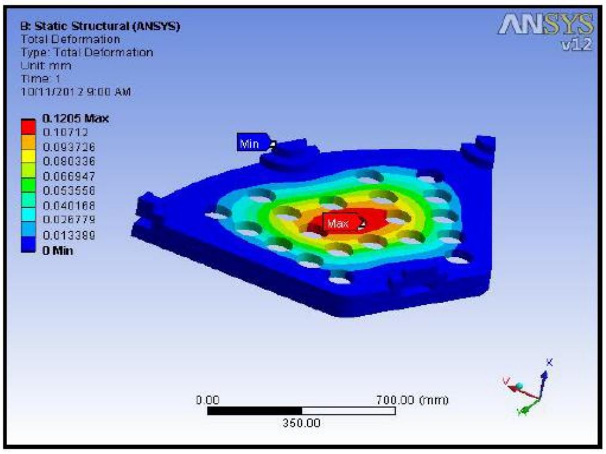


Figure 6.2: Maximum Deformation for Case A (Tetra Hedron)

6.1.3 Case B (Considering Hydrostatic Test Pressure & Standard Earth Gravity)

In Case B the conditions are same as Case A, only change is that the Standard Earth Gravity (9.8066 m/s2) is applied on filter plate assembly. And got the different values of Von Mises Stresses & Deformations for Different values of Element size. And two different meshing control methods are used to meshing the Filter Sheet Plate Assembly viz. - Tetrahedron meshing method & Hex- Dominant meshing method.

Filter Plate Pattern Used: Original Pattern as it is.

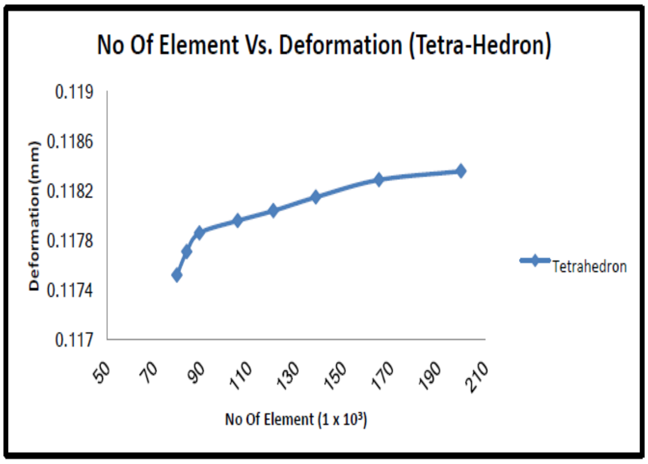
Sub steps: 5

Standard Earth Gravity: 9.8066 m/s2

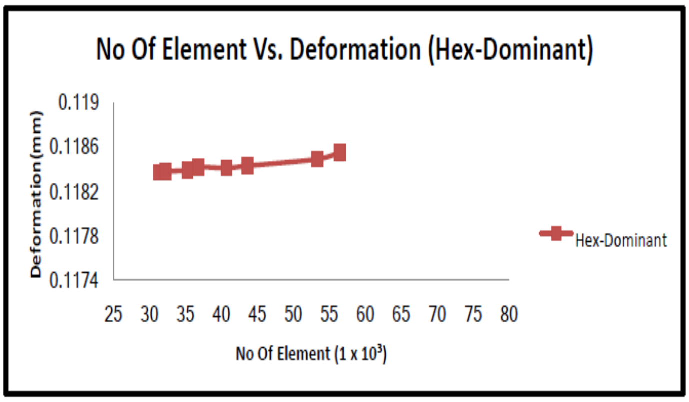
Meshing Type	Element Size(mm)	No of Element (1×10^3)	Deformation(mm)	Von-Mises Stress(Mpa)	Time (Sec)
Tetrahedron	21	79.542	0.11752	30.895	536
	20	83.712	0.11771	31.841	517
	19	89.077	0.11786	32.585	548
	18	105.244	0.11796	32.527	752
	17	120.35	0.11804	33.535	1890
	16	138.313	0.11815	33.304	929
	15	**165.034**	**0.11829**	**33.955**	**1559**
	14	199.736	0.11836	35.081	1559
Hex-Dominant	20	31.306	0.11837	32.162	452
	19	32.217	0.11838	32.69	448
	18	35.267	0.11839	32.941	543
	17	36.807	0.11842	33.169	480
	16	40.694	0.11841	33.861	741
	15	**43.606**	**0.11843**	**33.91**	**1559**
	14	53.296	0.11849	34.489	680
	13	56.508	0.11855	34.892	1062

Table 6-2: Result Table for Case B

Graph For Case B [no element VS Deformation]:



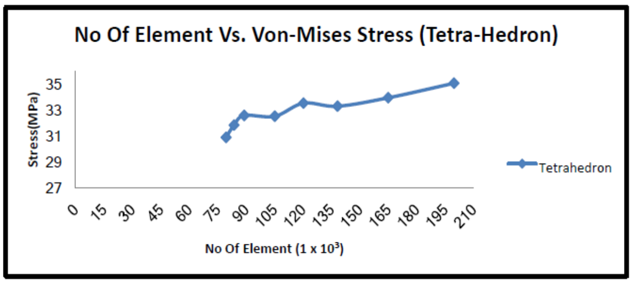
Graph 6-5: Graph for Case B of Tetra Hedron (No of Element vs. Deformation)



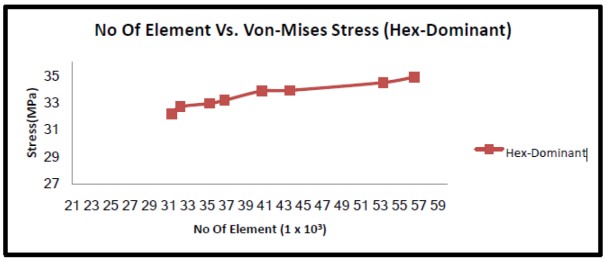
Graph 6-6: Graph for Case B of Hex Dominant (No of Element vs. Deformation)

From the graph of No of element vs. deformation of the filter plate assembly in case B, it is clear that, the graph is converged after the element size 15(no of element for tetrahedron = 165034 & no of element for Hex dominant = 43606). So that the deformation value for the Non-Linear Static Analysis using tetrahedron method meshing is 0.11829 mm at element size 15 mm and deformation value by using the Hex-dominant method meshing is 0.11843 mm at element size 15 mm.

Graph [no of Elements VS von-mises stress]:



Graph 6-7: Graph for Case B of Tetra Hedron (No of Element Vs. Von-Mises Stress)



Graph 6-8: Graph For Case B of Hex Dominant (No of Element Vs. Von-Mises Stress)

From the graph of No of element vs. Von-Mises stress of the filter plate assembly in case A, it is clear that, the graph is converged after the element size 15(no of element for tetrahedron = 165034 & no of element for Hex dominant = 43606). So that the Von-Mises value for the Non-Linear Static Analysis using tetrahedron method meshing is 33.955 MPa at element size 15 mm and Von-Mises value by using the Hex-dominant method meshing is 33.910 MPa at element size 15 mm.

6.1.4 Analysis Snapshots for Case B (Considering Hydrostatic Test Pressure & Standard Earth Gravity)

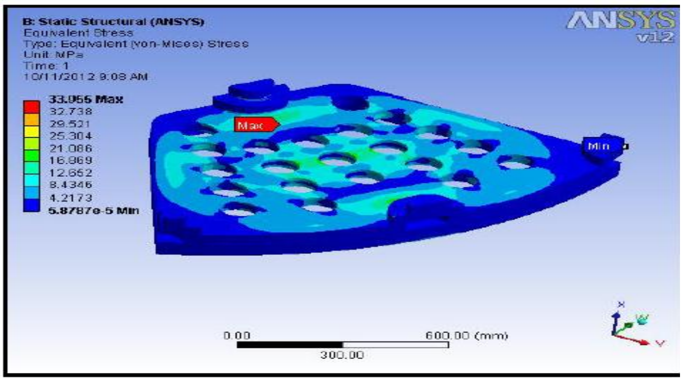


Figure 6.3: Maximum Stress for Case B (Tetra Hedron)

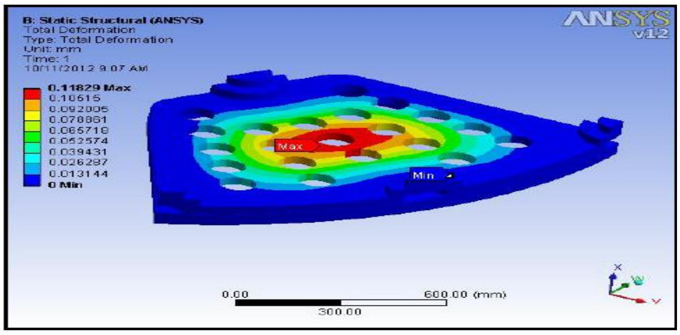


Figure 6.4: Maximum Deformation for Case B (Tetra Hedron)

6.1.5 Case C (Considering Back Pressure only)

In Case C the back pressure of 0.08 Mpa is applied instead of Hydrostatic test pressure. The direction of the back pressure is applied in the opposite direction of the applied Hydrostatic test pressure. And got the different values of Von Mises Stresses & Deformations for Different values of Element size. And two different meshing control methods are used to meshing the Filter Sheet Plate Assembly viz. - Tetrahedron meshing method & Hex-Dominant meshing method.

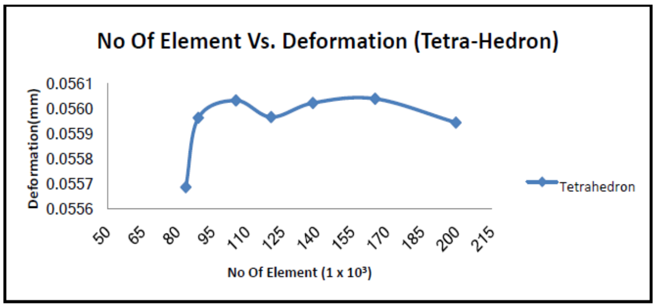
Filter Plate Pattern Used: Original Pattern as it is.

Sub steps: 5

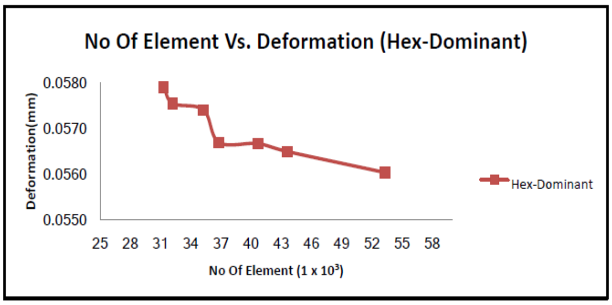
Meshing Type	Element Size(mm)	No of Element (1×10^3)	Deformation (mm)	Von-Mises Stress(Mpa)	Time (Sec)
Tetrahedron	20	83.712	0.055687	15.735	2227
	19	89.077	0.055963	11.48	3097
	18	105.244	0.056032	11.684	4839
	17	120.35	0.055966	14.542	5010
	16	138.313	0.056022	14.978	5492
	15	165.034	0.056039	14.712	5879
	14	199.736	0.055944	14.604	6013
Hex-Dominant	20	31.306	0.057895	15.945	2109
	19	32.217	0.057535	15.762	2228
	18	35.267	0.057397	15.531	2839
	17	36.807	0.056684	14.185	3100
	16	40.694	0.056662	13.814	3945
	15	43.606	0.056486	12.888	4143
	14	53.296	0.056029	12.213	4575

Table 6-3: Result Table for Case C

Graph For Case C [no element VS Deformation]:



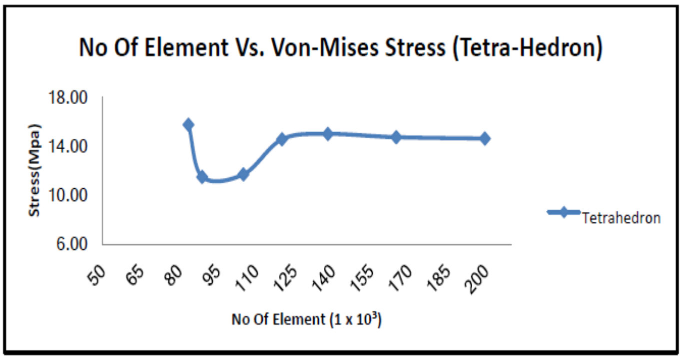
Graph 6-9: Graph for Case C of Tetra Hedron (No of Element vs. Deformation)



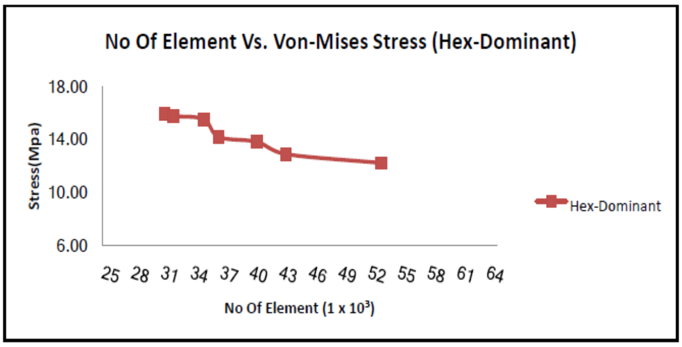
Graph 6-10: Graph for Case C of Hex- Dominant (No of Element vs. Deformation)

From the graph of No of element vs. deformation of the filter plate assembly in case C, it is clear that, the graph is converged after the element size 14mm (no of element = 199736) in case of tetrahedron type meshing method and deformation value is 0.055944 mm. And similarly in case of Hex-dominant meshing method, the graph is converged after the element size 18mm (no of element =35267). So that the deformation value for the Non-Linear Static Analysis using Hex-Dominant meshing method is 0.057397 mm.

Graph [no of Elements VS von-mises stress]:



Graph 6-11: Graph for Case C of Tetra Hedron (No of Element vs. Von-Mises Stresses)



Graph 6-12: Graph for Case C of Hex Dominant (No of Element vs. Von-Mises Stresses)

From the graph of No of element vs. Von-Mises stress of the filter plate assembly in case C, it is clear that, the graph is converged after the element size 14mm (no of element = 199736) in case of tetrahedron type meshing method and stress value is 14.604 MPa. And similarly in case of Hex-dominant meshing method, the graph is converged after the element size 18mm (no of element = 35267). So that the stress value for the Non-Linear Static Analysis using Hex-Dominant meshing method is 15.531MPa.

6.1.6 Analysis Snapshots for Case C (Considering Back Pressure only)

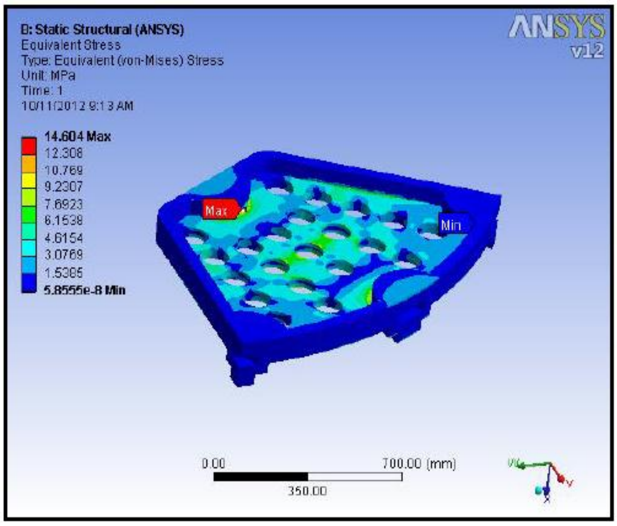


Figure 6.5: Maximum Stress for Case C (Tetra Hedron)

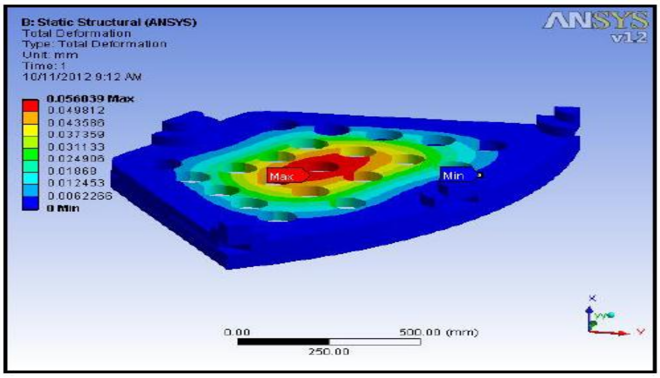


Figure 6.6: Maximum Deformation for Case C (Tetra Hedron)

6.1.7 Case D (Considering Back Pressure & Standard Earth Gravity)

In Case D the conditions are same as Case C, only change is that the Standard Earth Gravity is applied on filter plate assembly. And got the different values of Von Mises Stresses & Deformations for Different values of Element size. And two different meshing control methods are used to meshing the Filter Sheet Plate Assembly viz. - Tetrahedron meshing method & Hex- Dominant meshing method

Filter Plate Pattern Used: Original Pattern as it is.

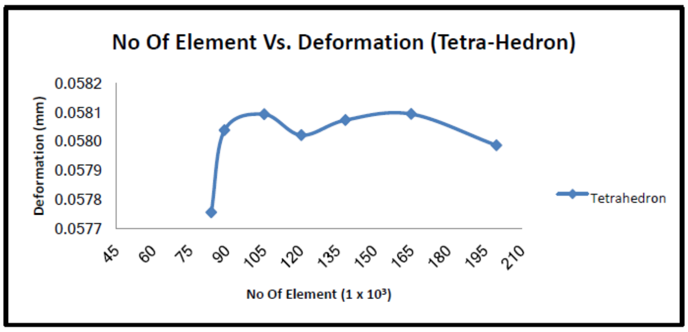
Sub steps: 5

Standard Earth Gravity: 9.8066 m/s2

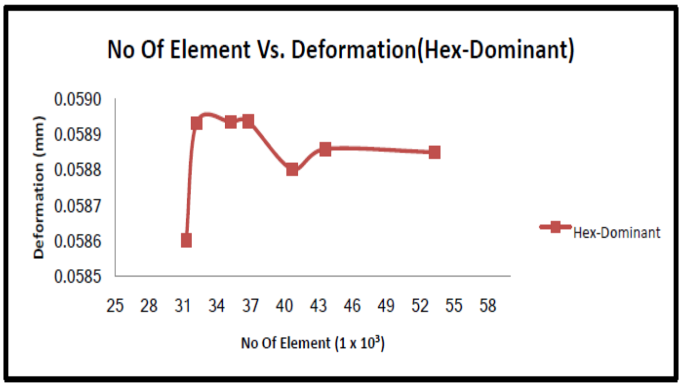
Meshing Type	Element Size(mm)	No of Element (1 x 10³)	Deformation (mm)	Von-Mises Stress(Mpa)	Time (Sec)
Tetrahedron	20	83.712	0.057756	16.307	681
	19	89.077	0.058039	11.896	605
	18	105.244	0.058094	12.105	807
	17	120.35	0.058022	14.966	1168
	16	138.313	0.058074	14.889	1580
	15	165.034	0.058095	14.35	1813
	14	199.736	0.057987	15.129	1813
Hex-Dominant	20	31.306	0.058603	17.75	548
	19	32.217	0.058932	15.49	461
	18	35.267	0.058935	13.329	688
	17	36.807	0.058936	13.421	863
	16	40.694	0.058802	13.546	883
	15	43.606	0.058859	13.284	813
	14	53.296	0.05885	13.23	1220

Table6-4: Result Table for Case D

Graph For D [no element VS Deformation]:



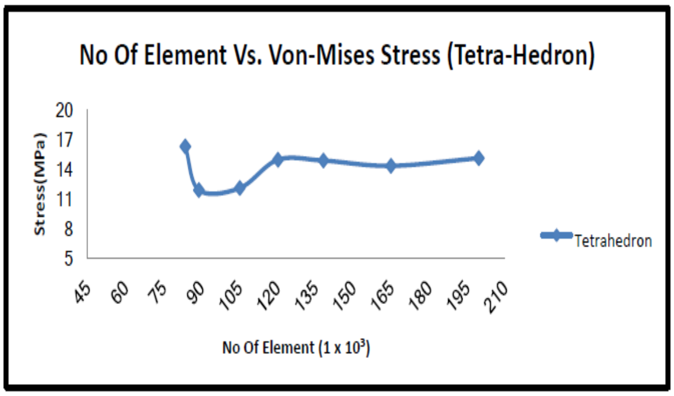
Graph 6-13: Graph for Case D of Tetra Hedron (No of Element vs. Deformation)



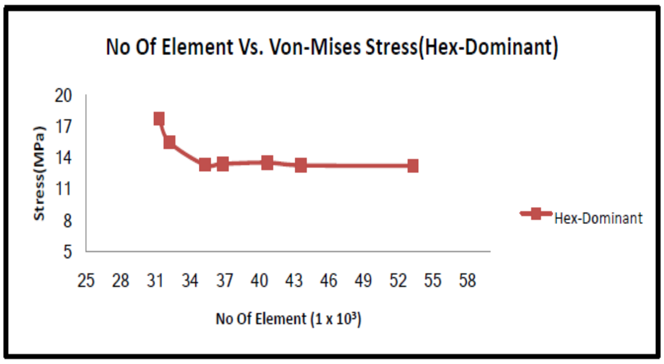
Graph 6-14: Graph for Case D of Hex Dominant (No of Element vs. Deformation)

From the graph of No of element vs. deformation of the filter plate assembly in case D, it is clear that, the graph is converged after the element size 15 mm (no of element = 165034) in case of tetrahedron type meshing method and deformation value is 0.058095 mm. And similarly in case of Hex-dominant meshing method, the graph is converged after the element size 14mm (no of element =53296). So that the deformation value for the Non-Linear Static Analysis using Hex-Dominant meshing method is 0.05885 mm.

Graph [no of Elements VS von-mises stress]:



Graph 6-15: Graph for Case D of Tetra Hedron (No of Element vs. Von-Mises Stresses)



Graph 6-16: Graph for Case D of Hex Dominant (No of Element vs. Von-Mises Stresses)

From the graph of No of element vs. Von-Mises stress of the filter plate assembly in case D, it is clear that, the graph is converged after the element size 15mm (no of element = 165034) in case of tetrahedron type meshing method and stress value is 14.350 MPa. And similarly in case of Hex-dominant meshing method, the graph is converged after the element size 14mm (no of element =53296). So that the stress value for the Non-Linear Static Analysis using Hex-Dominant meshing method is 13.23MPa.

6.1.8 Analysis Snapshots for Case D (Considering Back Pressure & Standard Earth Gravity)

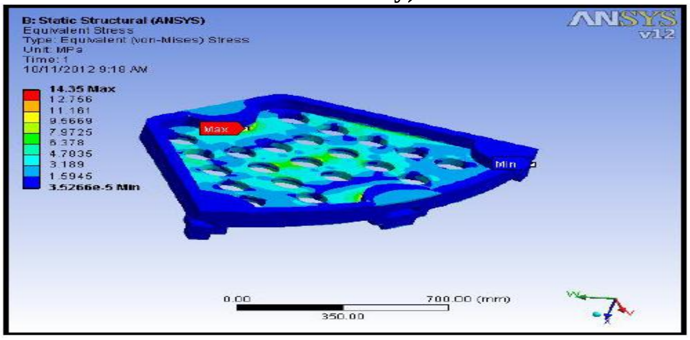


Figure 6.7: Maximum Stress for Case D(Tetra Hedron)

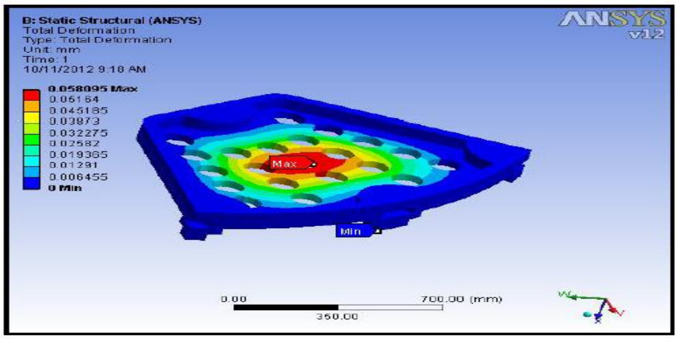


Figure 6.8: Maximum Deformation for Case D (Tetra Hedron)

From the above analysis of case A, B, C & D it is clear that,

1. Applying Hydrostatic Test Pressure of 0.175Mpa,

Case B is the Maximum von-mises stress & deformation is found.

σ = 33.955 Mpa for Tetrahedron method meshing

σ =33.91 Mpa for Hex-Dominant method meshing

At e

δ = 0.11829 mm for Tetrahedron method meshing

δ = 0.11843 mm Hex-Dominant method meshing

At e

2. Applying Back pressure of 0.08 Mpa

Case C is the Maximum von-mises stress & deformation is found.

σ = 14.604 Mpa for Tetrahedron method meshing at element size 14mm

σ = 15.531 Mpa for Hex-Dominant method meshing at element size 18mm.

δ = 0.055944 mm for Tetrahedron method meshing at element size 14mm.

δ = 0.11843 mm Hex-Dominant method meshing at element size 18mm.

So taking element size 15 mm for further analysis from the case B for tetrahedron method meshing. Also taking 14mm element size for tetrahedron from the Case C

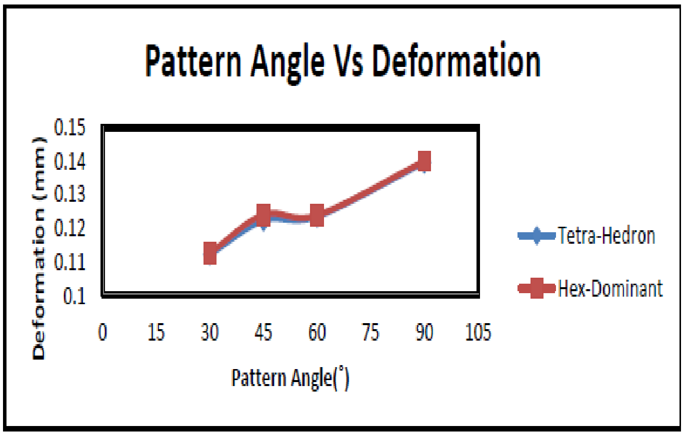
6.2 Analysis of Filter Sheet Plate by Changing the Pattern of Filter Tube Hole.

Now analysis is done by changing the pattern of filter plate. There are four types of pattern viz. moving the tube holes in 30°, 45°, 60° and 90° .Analysis is done by considering Case B. And taking element size constant i.e. - 15mm.

Meshing Type	Pattern Angle (°)	Element Size (mm)	No Of Element (1×10^3)	Deformation (mm)	Von-Mises Stress (Mpa)	Time (sec)	No Of Candles
Tetra-Hedron	original	15	165.034	0.11829	33.955	1559	21
	30		166.101	0.11205	34.306	1523	20
	45		162.896	0.12218	35.658	2005	23
	60		**156.037**	**0.12337**	**30.228**	**8212**	**30**
	90		157.224	0.13943	38.286	8059	28
Hex-Dominant	original	15	45.603	0.11862	33.91	708	21
	30		43.734	0.1125	37.17	583	20
	45		44.096	0.12394	39.819	614	23
	60		**42.131**	**0.12374**	**32.284**	**911**	**30**
	90		44.205	0.13985	40.428	644	28

Table 6-5: Result Table of Filter Plate Pattern Changing Analysis

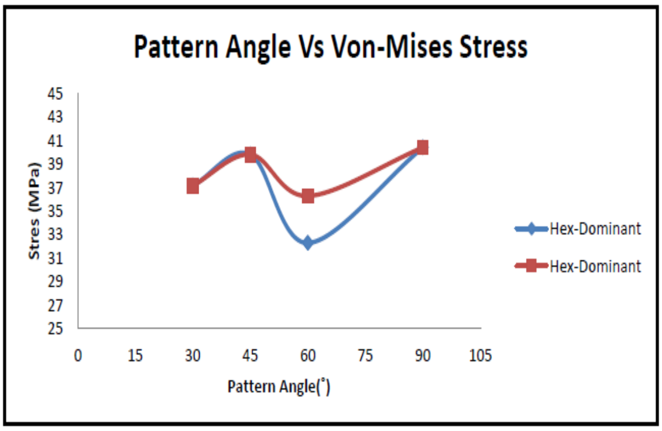
Graph [Pattern Angle Vs Deformation]:



Graph 6-17: Graph for Pattern Angle Vs Deformation [Tetra Hedron]

From the graph of pattern angle Vs Deformation it is conclude that the deformation value is minimum at the 60° pattern plate analysis. So that the 60° pattern filter plate is safe plate.

Graph [Pattern Angle VS von-mises stress]:



Graph 6-18: Graph for Pattern Angle Vs von mises stress

From the graph of pattern angle Vs Stress it is conclude that the Von-Mises stress value is minimum at the 60° pattern plate analysis. So that the 60° patterns filter plate is safe plate.

6.2.1 Analysis snapshots for Analysis of Filter Sheet Plate by Changing the Pattern

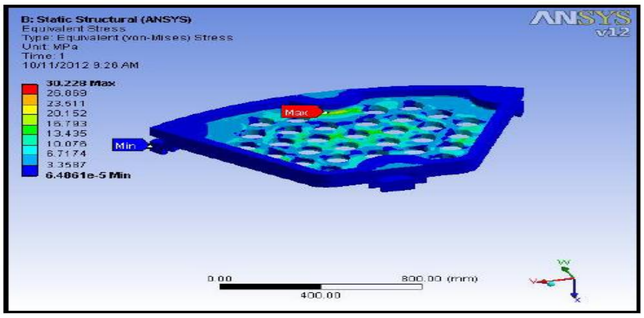


Figure 6.9: Maximum Stress for analysis of changing the filter sheet plate pattern

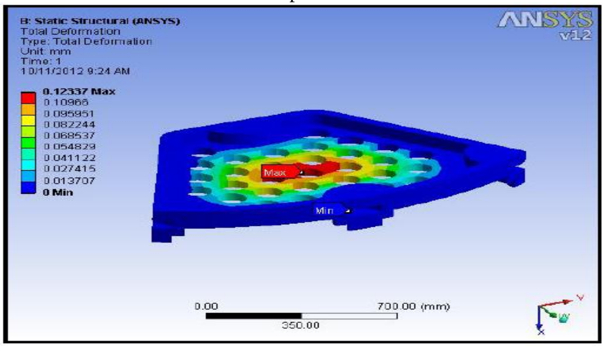


Figure 6.10: Maximum Deformation for analysis of changing the filter sheet plate pattern

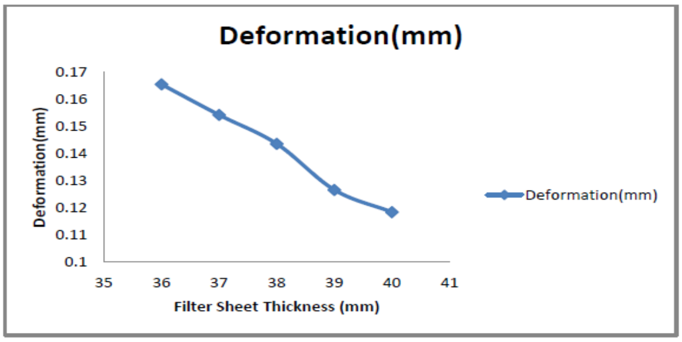
6.3 Analysis of Filter Sheet Plate by Changing the Thickness of Filter Plate.

Now analysis is done by changing the thickness of filter sheet plate. The existing thickness of the Filter sheet plate is 40 mm. Now the Analysis is done by decreasing the thickness of the filter sheet plate. Analysis is done by considering Case B. And taking element size constant i.e. - 15mm.

Meshing Type	Filter Sheet Thickness (mm)	No Of Element (1×10^3)	Deformation (mm)	Von-Mises Stress(Mpa)	Time(sec)
Tetra-Hedron	40	165.034	0.11829	33.955	1559
	39	161.024	0.12649	33.482	2098
	38	158.389	0.14353	34.287	2030
	37	156.72	0.15422	35.596	3302
	36	154.963	0.16553	44.093	3461

Table 6-6: Result Table of Filter Sheet Plate Thickness Changing Analysis:

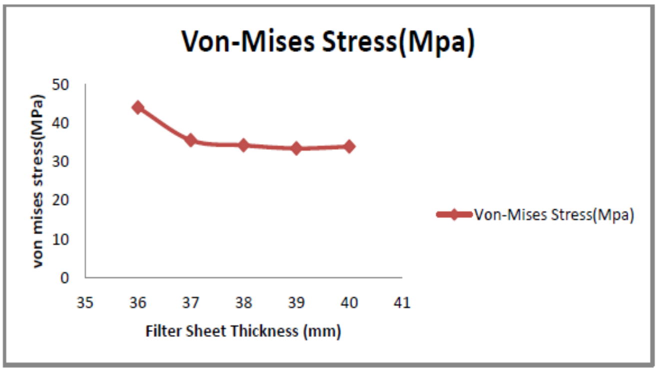
Graph [Filter Sheet Plate Thickness Vs Deformation]:



Graph 6-19: Graph for Filter Sheet Plate Thickness vs. Deformation

From the graph of Deformation Vs Filter Sheet Thickness it is conclude that as the thickness of filter sheet increases the deformation in Filter Sheet Plate Decreases.

Graph [Filter Sheet Plate Thickness VS von-mises stress]:



Graph 6-20: Graph for Filter Sheet Plate Thickness Vs von mises stress

From the Graph of Von-Mises Stress Vs Filter Sheet Thickness it is conclude that as the thickness of the filter plate increases the stress of the plate decreases. Also it is seen that after the 39 mm thickness of plate the graph is converged. So value of stress is 33.955 Mpa so 40mm of thickness of filter sheet plate is taking for analysis.

6.3.1 Analysis snapshots for Analysis of Filter Sheet Plate by Changing the Thickness of Filter Plate.

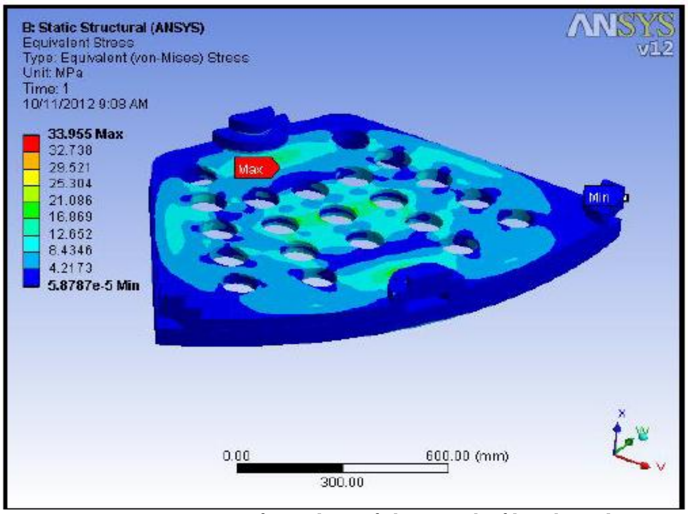


Figure 6.11: Maximum Stress for analysis of changing the filter sheet plate thickness

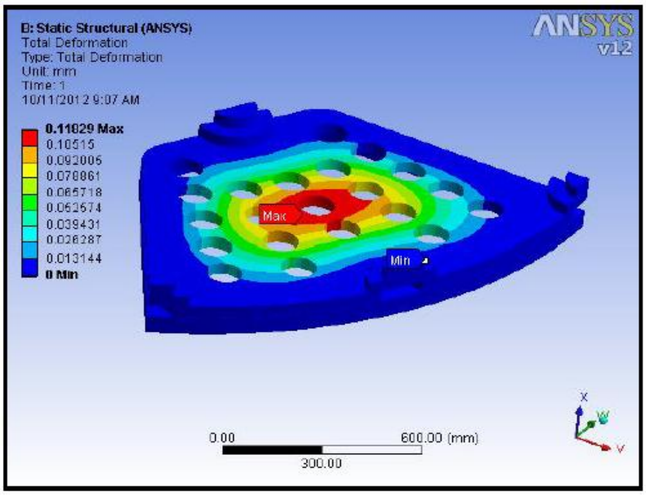


Figure 6.12: Maximum Deformation for analysis of changing the filter sheet plate thickness

6.4 Analysis of Complete Filter Sheet Plate Assembly

Now the Analysis of Complete Filter Sheet Plate assembly is done by Extreme condition. These Conditions are selected according to the analysis of 1/6th of Filter Sheet Plate Assembly. These conditions are as follows.

a. Engineering DATA

- Material : Structural Steel
- Mass : 424.68Kg
- Density : 7850 Kg/ m^3
- Volume : 5.433E7 m^3
- Young's Modulus : 2E11 Pa
- Poisson's ratio : 0.3
- Yield strength : 2.5E8 Pa

b. Meshing Control Used: Tetrahedron Meshing

c. Element size used : 30mm

d. Pattern Used: 60° Filter tube hole pattern Filter Sheet Pattern

e. Hydrostatic Test Pressure: 0.175 Mpa

f. Standard Earth Gravity: 9.8066 m/s2

6.4.1 Complete Filter Sheet Plate Assembly.

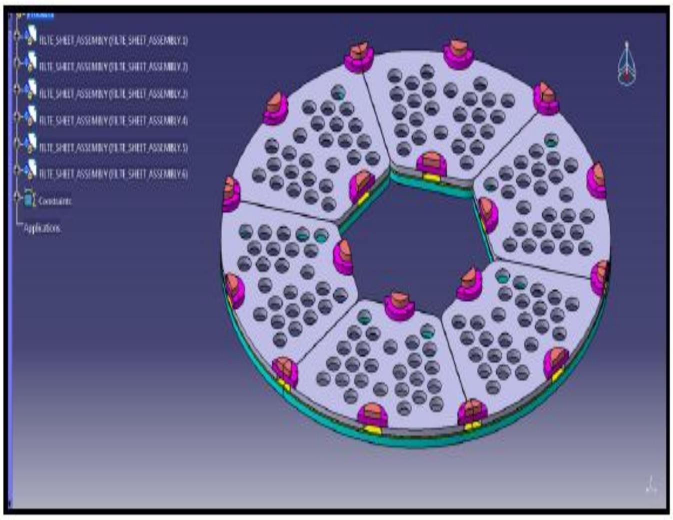


Figure 6.13: 3-D drawing of Complete Filter sheet Plate Assembly

Figure 6.1 shows the 3-D model of complete Filter Sheet Plate assembly of 60° Filter tube hole pattern. This model is modeled in CATIA V5 R18.

6.4.2 Model Imported from the CATIA to the ANSYS WORKBENCH

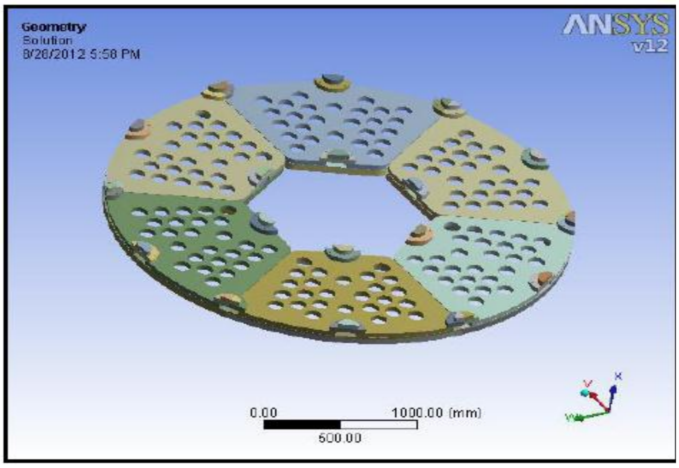


Figure 6.14: Imported model of Complete Filter sheet Plate Assembly from CATIA

6.4.3 Tetrahedron Meshed model of Complete Filter Sheet Plate Assembly.

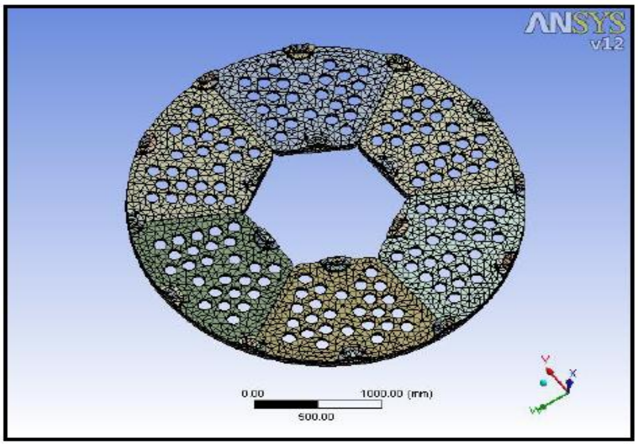


Figure 6.15: Meshed Model of complete Filter sheet Plate Assembly

6.4.4 Boundary Conditions Applied on Filter Sheet Plate Assembly

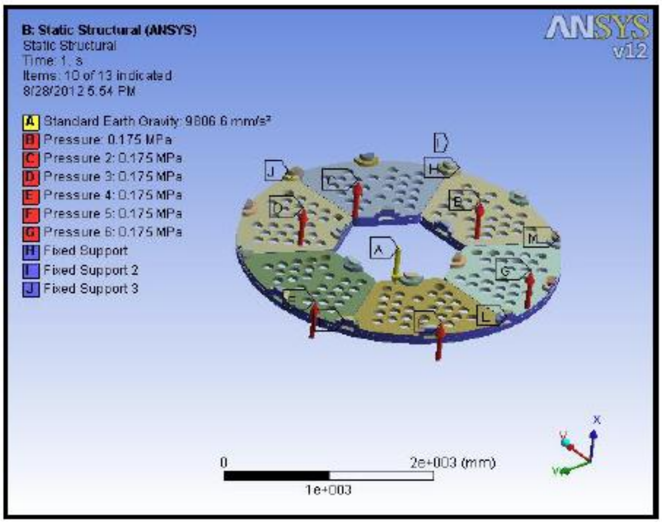


Figure 6.16: Boundary condition applied on Full Filter sheet Plate Assembly

Figure 6.3 shows the boundary condition applied on the full filter sheet plate assembly. The outer surface is welded to the shell and hydrostatic test pressure of 0.175 Mpa is applied on the filter plate in the X direction. Also the standard earth gravity is applied to the full filter sheet plate assembly. And then solved for the maximum deformation and maximum von-mises stress.

Meshing Type	Filter Sheet Assembly Type	Element size (mm)	No Of Element (1×10^3)	Deformation (mm)	Von-Mises Stress(Mpa)	Time (sec)
Tetra-Hedron	Case B-60 degree pattern	30	236.1	0.11914	36.176	14832

Table 6-7: Result Table for Complete Filter Sheet Plate Analysis.

6.4.5 Analysis Snapshots for Analysis Of Complete Filter Sheet Plate Assembly.

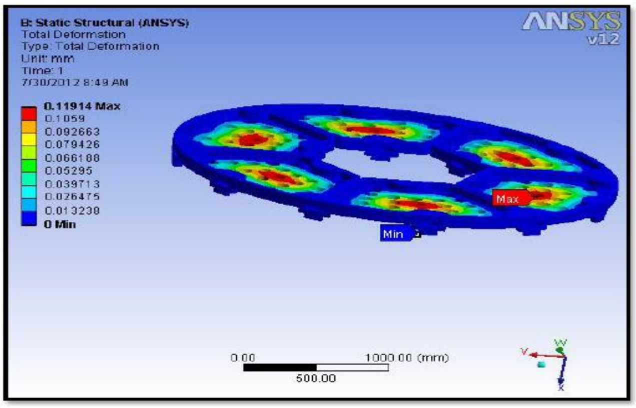


Figure 6.17: Maximum Deformation In complete filter sheet Plate Assembly

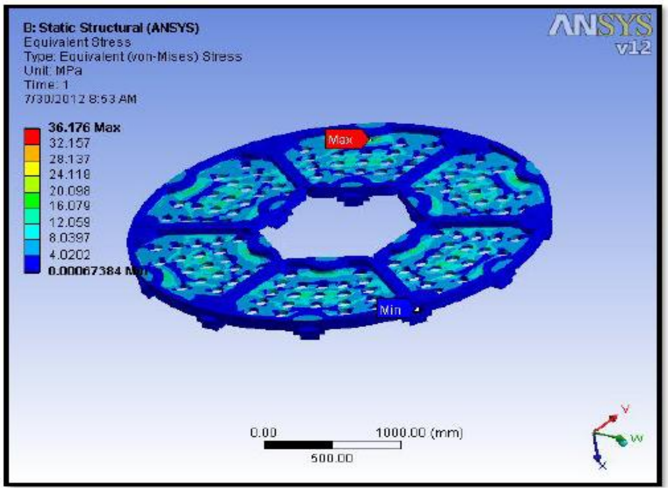


Figure 6.18: Maximum Von Mises Stress In complete filter sheet Plate Assembly.

VII. CONCLUSION AND FUTURE SCOPE

8.1 Conclusion

From the above non-linear static structural analysis of Filter Sheet Plate assembly and graphs plotted accordance with results, it is conclude that,

- The stresses induced by applying given boundary condition in all the cases of analysis are less than Design Stress value which 250 Mpa is considering Factor of Safety 5.
- The maximum Von Mises stresses after shape optimization were found to be less in the shape optimized model than those in previously proposed model.
- After carrying out structural analysis it is conclude that the 60° pattern of hole tube of filter sheet plate is maximum safe.
- After carrying out analysis by changing the thickness of the filter sheet plate it is conclude that, the 40 mm thickness is safe.
- The difference between the deformation of filter sheet plate in experimental result and analytical result by ANSYS is as follows

  Deformation by Experimental : 0.141 mm

  Deformation by ANSYS : 0.11914 mm

  Difference : 0.02186 mm

- The maximum value of stress in this case is found out on the filter plate as shown in fig. It is also less than the yield strength of the material considering the factor of safety. For all other operating conditions stress plots are same but values are less than hydro test operating condition. Hence the filter sheet assembly is safe in all operating condition after the optimization.

8.2 Future Scope

Finite Element Analysis of Filter Sheet Plate was carried out by changing the filter tube hole pattern and thickness of the filter sheet plate in present work. Currently the filter sheet plate was designed but in future we can model out the outer side of shell of the pressure vessel.

REFERENCES

1. David H., Gene M, Mark G. (1998), “Finite Element Analysis of pressure vessel”.

2. Wang Y, Lu Y .Pan H. (18 January 2008), “Failure analysis of a hydro-processing reactor”, Engineering Failure Analysis’, Vol 16, P.P. 11–18.

3. J.E. Meyer, (August 1996), “Pressure Vessel Stress Analysis”.

4. Brian Vicich; Craig Ryan; Kevin Meredith, “Linear vs. Non-Linear Contact Analysis”, Samtec, Inc. 2007.

5. R.C. Carbonari, E.C.N. Silva, P.A. Muñoz-Rojas, E.Q. Andrade, G.H. Paulino, K. Nishimoto (11 May 2011), “Design of pressure vessels using shape optimization”, International Journal of Pressure Vessels and Piping 88, 198 – 212.

6. Z. Sanal, (5 December 2000), “Nonlinear analysis of pressure vessels: some examples”, International Journal of Pressure Vessels and Piping 77, 705 – 709.

7. Shafique M.A. Khan, (8 March 2010), “Stress distributions in a horizontal pressure vessel and the saddle supports”, International Journal of Pressure Vessels and Piping 87, 239 – 244.

8. K. Magnucki, W. Szyc, J. Lewin´ski, K. Magnucki, (11 September 2002), “Minimization of stress concentration factor in cylindrical pressure vessels”, International Journal of Pressure Vessels and Piping 79, 841–846.

9. Shu-Ho Dai, (11 March 1998), “Methodology for predicting the occurrence of failure events for pressure vessels used in the process industry”, International Journal of Pressure Vessels and Piping 75,221–228.

10. Dong-xia Liu, Li Liang, Ming Li, Dong-xia Liu, (2010), “Nonlinear finite element analysis of mechanical characteristics on CFRP composite pressure vessel”, IOP Conf. Series: Materials Science and Engineering 10.

11. Yogesh Borse, Avadesh K. Sharma, (May-Jun 2012), “Modelling of Pressure Vessels with different End Connections using Pro Mechanica”, International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 www.ijera.com Vol. 2, Issue 3, pp.1493-1497.

12. V. G. Ukadgaonker, P. A. Kale, Mrs. N. A. Agnihotri, R. Shanmuga Babu, (9 February 1995), “Review of analysis of tube sheets”, International Journal of Pressure Vessels and Piping 61, 219-297.

13. W. J. O'Donnell, B. F. Langer, (August 1962), “Design of Perforated Plates”, ASME Journal of Engineering for Industry, Trans. Vol. 84, Paper No 61-WA-115.

14. Tong LIU, Ke WANG, Qi-Wu DONG, Min-Shan LIU, (25 May 2009), “Hydroelastic natural vibrations of

perforated plates with cracks", Procedia Engineering 1, 129–133.

15. Klaus Jiirgen Bathe, Arthur P. Cimento, (21 March 1979), "Some Practical Procedures for the Solution of Nonlinear Finite Element Equations", Computer Methods in Applied Mechanics and Engineering 2, 59-85 North-Holland Publishing Company.

16. R.B. Tait, J. Press, (12 February 2000), "An experimental study of the residual stresses, and their alleviation, in tube to tube-sheet welds of industrial boilers", Engineering Failure Analysis 8, 15 – 27.

17. Khosrow Behseta, Donald Mackenzie, Robert Hamilton, (5 December 2011), "Plastic load evaluation for a fixed tube sheet heat exchanger subject to proportional loading", International Journal of Pressure Vessels and Piping 92, 11-18.

18. O.R. Nandagopan, S. Ranjith Kumar, M.R. Rajesh, K. Manoharan, and C.G. Nandakumar, (06 July 2012), "Plastic load evaluation for a fixed tube sheet heat exchanger subject to proportional loading", Defence Science Journal, Vol. 62, No. 4, pp. 248-251.

19. Dassault Systèmes SolidWorks Corp., "Understanding Nonlinear Analysis".

20. Dr D H Nash, (February 2010), "Finite Element Analysis Of Horizontal Filter Vessel With Stiffened Central Stayed Dividing Plate".

21. Jonathan Fraser, "FEA Summary".

22. Book- "Pressure Vessel Design", J. F. Harvey.

23. Book- "Pressure Vessel Design Manual", Dennis Moss.

24. An International code 2007 ASME Boiler & Pressure Vessel Code, July 1, 2008.

25. ANSYS 12.0 Work Bench Help