ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

# BUCKLING AND FINITE ELEMENT ANALYSIS OF CYLINDRICAL PRESSURE VESSEL

G. Nomukumar
MTech Student
Department of Mechanical Engineering
KSRM Engineering College, Kadapa,

P. Sreenivas, M.Tech
Assistant professor
Department of Mechanical Engineering
KSRM Engineering College, Kadapa,

***ABSTRACT:*** ***Cutouts in cylindrical shell-type components are unavoidable in the construction of aerospace structures. This fact is significant because the structural failure of these components usually begins near the cutout because of high stress concentrations that initiate the formation of cracks. Hence, a cutout can trigger a local failure at a load level lower than the global failure load of a corresponding shell without a cutout. As a result, preliminary-design sizing of a cylindrical shell with a cutout is often based on the magnitude of the stress concentrations near the cutout. Therefore, an accurate assessment of the stress concentrations in a given shell subjected to various types of loading and support conditions is essential to the development of safe and reliable designs. Moreover, validated special-purpose analysis tools that enable rapid parametric studies would be very valuable to structural designers and for the development of new design criteria and design concepts. In this thesis, static and buckling analyses are to be done for determining the stress and deformation fields in a thin laminated-composite cylindrical shell with an elliptical without cutout and elliptical cutout is presented. The materials used are Steel, S2 Glass Epoxy and Kevlar. The analysis is done using solid element and shell element. 5 and 11 layers are used as layer stacking for composite materials S2 Glass and Kevlar. Modeling is done in Pro/Engineer and analysis is in Ansys.***

## I.INTRODUCTION

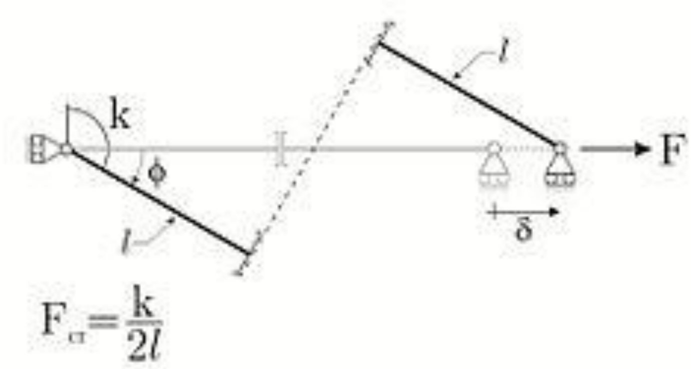
Shell structures are widely used structural components in engineering designs. Basically, a shell structure is a three – dimensional structure which is thin in one direction and long in the other two directions. Because of this geometric aspect, they are thin, light and they span over large areas. Although they are thin, they carry applied loads effectively by means of their curvatures. The objective of the design is to make the shell as thin as possible in order to have a light and low cost, but a functional structure. However shells can be extremely sensitive to imperfections and changes in the thickness and boundary conditions. A special type of shells is the cylindrical shell. Cylindrical shells are found in many applications in the aerospace and naval construction industries. They are often used as load – bearing structures for aircraft, rockets, submarines and missile bodies. Of course, the cylindrical shells used in those designs are stiffened to achieve better strength, stiffness and buckling characteristics The theory of structures tends to deal with a class of idealized mathematical models, stripped of many of the features that make them recognizable as useful object in engineering. Thus a beam is often idealized as a line endowed with certain mechanical properties, irrespective of whether it is a large bridge, an aircraft wing, or a flat spring inside a machine. In a similar way, the theory of shell structures deals, for example, with the “cylindrical shell” as an idealized entity: it is a cylindrical surface endowed with certain mechanical properties. The treatment is the same whether the actual structure under study is a gas-transmission pipeline, a grain storage silo, or a steam boiler.

## II.INTRODUCTION TO BUCKLING ANALYSIS

In science, buckling is a mathematical instability, leading to a failure mode. Theoretically, buckling is caused by a bifurcation in the solution to the equations of static equilibrium. At a certain stage under an increasing load, further load is able to be sustained in one of two states of equilibrium: an un deformed state or a laterally-deformed state .In practice, buckling is characterized by a sudden failure of a structural member subjected to high compressive stress, where the actual compressive stress at the point of failure is less than the ultimate compressive stresses that the material is capable of withstanding. For example, during earthquakes, reinforced

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

concrete members may experience lateral deformation of the longitudinal reinforcing bars. This mode of failure is also described as failure due to elastic instability. Mathematical analysis of buckling makes use of an axial load eccentricity that introduces a moment, which does not form part of the primary forces to which the member is subjected. When load is constantly being applied on a member, such as column, it will ultimately become large enough to cause the member to become unstable. Further load will cause significant and somewhat unpredictable deformations, possibly leading to complete loss of load-carrying capacity. The member is said to have buckled, to have deformed. A free-standing, vertical column, with density $\rho$, Young's modulus $E$, and radius $r$, will buckle under its own weight if its height exceeds a certain critical height: $h_{crit} = \left(\frac{9B^2}{4}\frac{EI}{\rho g \pi r^2}\right)^{1/3}$ where $g$ is the acceleration due to gravity, $I$ is the second moment of area of the beam cross section, and $B$ is the first zero of the Bessel function of the first kind of order -1/3, which is equal to 1.86635...




## III.INTRODUCTION TO S2 GLASS

High-strength glass, carbon or other advanced fibers are used in applications requiring greater strength and lower weight. High-strength glass is generally known as S-type glass in the United States, R-glass in Europe and T-glass in Japan. S-glass was originally developed for military applications in the 1960s, and a lower cost version, S-2 glass, was later developed for commercial applications. High-strength glass has appreciably higher amounts of silica oxide, aluminum oxide and magnesium oxide than E-glass. S-2 glass is approximately 40-70% stronger than E-glass.

### PROPERTIES OF S2 GLASS

| **Physical Properties** | **Metric** |
|---|---|
| Density | 2.46 g/cc |
| **Mechanical Properties** | **Metric** |
| Tensile Strength, Ultimate | 4890 MPa |
| Elongation at Break | 5.70 % |
| Modulus of Elasticity | 86.9 GPa |
| Poissons Ratio | 0.230 |
| Shear Modulus | 35.0 GPa |

## IV.INTRODUCTION TO KEVLAR

Kevlar is the registered trademark for a para-aramid synthetic fiber, related to other aramids such as Nomex and Technora. Developed by Stephanie Kwolek at DuPont in 1965, this high-strength material was first commercially used in the early 1970s as a replacement for steel in racing tires. Typically it is spun into ropes or fabric sheets that can be used as such or as an ingredient in composite material components. Currently, Kevlar has many applications, ranging from bicycle tires and racing sails to body armor because of its high tensile strength-to-weight ratio; by this it measures is 5 times stronger than steel.

### PROPERTIES of KEVLAR

| **Physical Properties** | **Metric** |
|---|---|
| Density | 1.44 g/cc |
| **Mechanical Properties** | **Metric** |
| Tensile Strength | 3000 MPa, |
| Ultimate Strength | 3620 MPa |

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

| | |
|---|---|
| Tenacity | 2.08 N/tex |
| Elongation at Break | 2.40 % |
| Tensile Modulus | 112 GPa |
| Poisson's Ratio | 0.360 |

## DESIGN OF ELLIPTICAL HEAD PARAMETERS

ASME design equation for elliptical head

t=PDK/(2SE-0.2P)

Thickness of elliptical head = t in mm

P=design pressure in N/mm$^2$

D=outer diameter in mm

Design Pressure (P) = 21 N/mm$^2$

Allowable Stress value(S) = 123 N/mm$^2$

Joint Efficiency (E) = 1

Stress concentration factor (k) =1/6 [2+ ( 〖a/b)$^2$ ]

Major axis length (a) =2724mm

Minor axis length (b) =771.34mm

k=1/6 [2+ ( 〖2724/771.34)$^2$ ]=0.62

t=(21×2724)/(2×123×1-0.2×21)=146.67≈147

Hoop stress (σ_t1) =(P×D)/(2×t)

= (21×2724)/(2×147) =195 mpa

Longitudinalstress(σ_t2) =(P×D)/(4×t)=(21×2724)/(4×147)= 97MPa

## INTRODUCTION TO CAD

**Computer-aided design** (**CAD**), also known as **computer-aided design and drafting** (**CADD**), is the use of computer technology for the process of design and design-documentation. Computer Aided Drafting describes the process of drafting with a computer. CADD software, or environments, provide the user with input-tools for the purpose of streamlining design processes; drafting, documentation, and manufacturing processes. CADD output is often in the form of electronic files for print or machining operations. The development of CADD-based software is in direct correlation with the processes it seeks to economize; industry-based software (construction, manufacturing, etc.) typically uses vector-based (linear) environments whereas graphic-based software utilizes raster-based (pixelated) environments.CADD environments often involve more than just shapes. As in the manual drafting of technical and engineering drawings, the output of CAD must convey information, such as materials, processes, dimensions, and tolerances, according to application-specific conventions. CAD may be used to design curves and figures in two-dimensional (2D) space; or curves, surfaces, and solids in three-dimensional (3D) objects. CAD is an important industrial art extensively used in many applications, including automotive, shipbuilding, and aerospace industries, industrial and architectural design, prosthetics, and many more. CAD is also widely used to produce computer animation for special effects in movies, advertising and technical manuals. The modern ubiquity and power of computers means that even perfume bottles and shampoo dispensers are designed using techniques unheard of by engineers of the 1960s. Because of its enormous economic importance, CAD has been a major driving force for research in computational geometry, computer graphics (both hardware and software), and discrete differential geometry. The design of geometric models for object shapes, in particular, is often called *computer-aided geometric design* (*CAGD*).Current computer-aided design software packages range from 2D vector-based drafting systems to 3D solid and surface modellers. Modern CAD packages can also frequently allow rotations in three dimensions, allowing viewing of a designed object from any desired angle, even from the inside looking out. Some CAD software is capable of dynamic mathematic modeling, in which case it may be marketed as **CADD** — *computer-aided design and drafting*.CAD is used in the design of tools and machinery and in the drafting and design of all types of buildings, from small residential types (houses) to the largest commercial and industrial structures (hospitals and factories).CAD is mainly used for detailed engineering of 3D models and/or 2D drawings of physical components, but it is also used throughout the engineering process from conceptual design and layout of products, through strength and dynamic analysis of assemblies to definition of manufacturing methods of components. It can also be used to design objects.

## INTRODUCTION TO PRO/ENGINEER

Pro/ENGINEER Wildfire is the standard in 3D product design, featuring industry-leading productivity tools that promote best practices in design while ensuring compliance with your industry and company standards. Integrated Pro/ENGINEER CAD/CAM/CAE solutions allow you to

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

design faster than ever, while maximizing innovation and quality to ultimately create exceptional products. Customer requirements may change and time pressures may continue to mount, but your product design needs remain the same - regardless of your project's scope, you need the powerful, easy-to-use, affordable solution that Pro/ENGINEER provides.

## PRO/ENGINEER WILDFIRE BENEFITS

- Unsurpassed geometry creation capabilities allow superior product differentiation and manufacturability
- Fully integrated applications allow you to develop everything from concept to manufacturing within one application
- Automatic propagation of design changes to all downstream deliverables allows you to design with confidence
- Complete virtual simulation capabilities enable you to improve product performance and exceed product quality goals
- Automated generation of associative tooling design, assembly instructions, and machine code allow for maximum production efficiency

Pro ENGINEER can be packaged in different versions to suit your needs, from Pro/ENGINEER Foundation XE, to Advanced XE Package and Enterprise XE Package, Pro/ENGINEER Foundation XE Package brings together a broad base of functionality. From robust part modeling to advanced surfacing, powerful assembly modeling and simulation, your needs will be met with this scalable solution. The main modules are **Part Design, Assembly Drawing, and Sheet Metal**

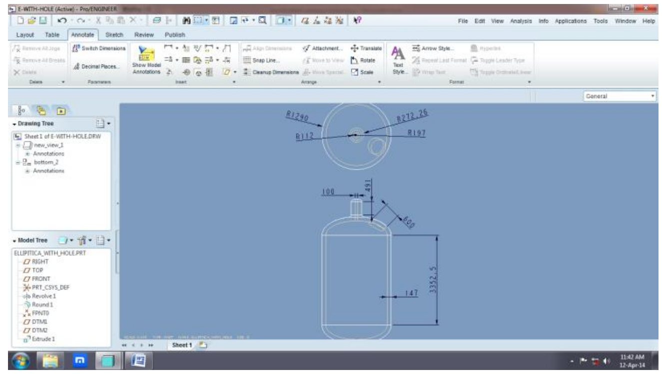


FIG 1

FIG 1 SHOWS PART DESIGN IN 2D

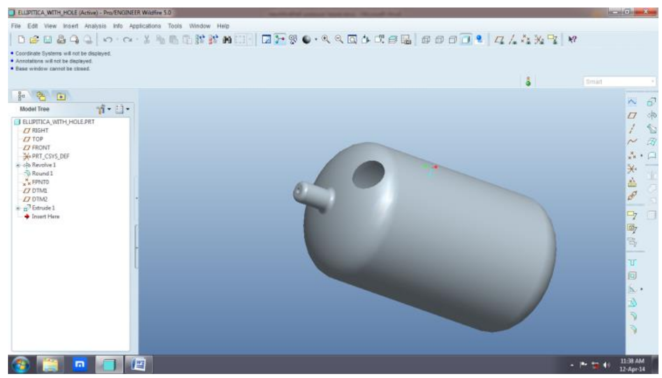


FIG 2

FIG 2 REPRESENTS ASSEMBLY

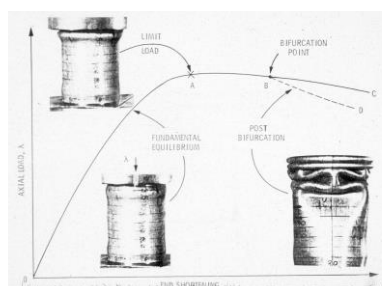
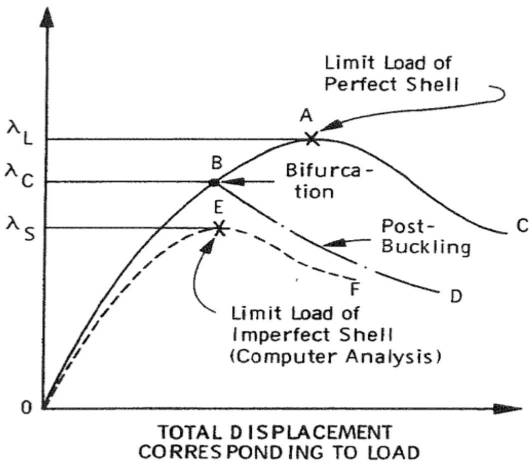



FIG 3

FIG 3 SHOWS APPLICATION OF LOAD ON CYLINDER

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal




THIS GRAPH SHOWS THE DEFORMATION OF CYLINDER WHEN THE LOAD WAS APPLIED

## FINITE ELEMENT ANALYSIS

Finite element analysis (FEA) is a computerized method for predicting how a component/assembly will react to environmental factors such as forces, heat, and vibration. Though it is called "analysis," in the product design cycle, it is used as a virtual prototyping tool to predict what is going to happen when the product is used. Finite element analysis, as related to the mechanics of solids, is the solution of a finite set of algebraic matrix equations that approximate the relationships between load and deflection in static analysis and velocity, acceleration, and time in dynamic analysis. The basic idea in the finite element method is to find the solution of a complicated problem by replacing it by a simpler one. Since the actual problem is replaced by a simpler one in finding the solution, we will be able to find only an approximate solution rather than the exact solution. The existing mathematical tools will not be sufficient to find the exact solution (and sometimes, even an approximate solution) of most of the practical problems.

Thus in the absence of any other convenient method to find even the approximate solution of a given problem, we have to prefer the **FINITE ELEMENT METHOD**. Moreover, in finite element method, it will often be possible to improve or refine the approximate solution by spending more computational effort.

### Advantages of Finite Element Analysis:

1. In contrast to other variation and residual approaches the finite element method does not require trial solutions, which apply to entire multi dimensional continuum.
2. The use of separate sub regions, or finite elements, for the trial solutions permits a greater flexibility in considering of complex shape.
3. Rather than requiring every trail solution to satisfy the boundary conditions, one prescribes the conditions after obtaining the algebraic equations for the assemblage.
4. As boundary conditions do not enter in to equations for the individual finite elements, one can use the same field variable for both internal and boundary elements.
5. The filed variable models need not be changed when the boundary conditions change.
6. The introduction of boundary conditions in to assembled equations is a relatively easy process. No special techniques or artificial device are necessary.
7. The finite element method not only accommodates complex geometry and boundary conditions, but also proven successful in representing various types of complicated material properties that are difficult to incorporate in the numerical methods.

### Limitations of Finite Element Analysis:

The finite element method does not accommodate few complex phenomena

**1.** Cracking and fracture behavior.
**2.** Contact problems.
**3.** Bond failures of composite materials.
**4**. Non-linear behavior with work softening.
**5.** It does not account for transient, unconfined seepage problems.
**6.** The finite element analysis has reached a high level of development as a solution technique. However, the method yields realistic results only if the coefficients or material parameters, which describe the basic phenomena, are available.

## GENERAL PROCEDURE OF FINITE ELEMENT METHOD

The solution of a continuum problem by the finite element method usually follows an orderly step-by-step process. The following steps show in general how the finite element method

## SELECT THE SOLUTION APPROXIMATION

The variation of the field variables in the problem (unknown) is approximated within each element by a polynomial, the field variables may be scalar (ex. Temperature) or a vector (displacement). Polynomials are usually used to approximate the solution over an element domain because they are easy to integrate and differentiate. The degree of polynomial depends on the number of nodes per element, the number of unknowns at each node and certain continuity requirements along element boundaries.

## ANSYS – AN OVERVIEW

ANSYS is finite element analysis software for developed by ANSYS INC for design and analysis of engineering problems. It is user friendly graphical user interface package. Many CAD Programmes have direct interfaces with the ANSYS program through software written by ANSYS.INC or by the CAD vendors. Translators for the programs like AutoCAD and Pro/Engineer are available from ANSYS.INC.

ANSYS finite element analysis software enables engineers to perform the following tasks.

1)Build computer models or transfer CAD models of structures, products, components or system.

2)Apply operating loads or other design performance conditions.

## PROCEDURE FOR ANSYS ANALYSIS:

Static analysis is used to determine the displacements, stresses, strains and forces in structures or components due to loads that do not induce significant inertia and damping effects. Steady loading in response conditions are assumed. The kinds of loading that can be applied in a static analysis include externally applied forces and pressures, steady state inertial forces such as gravity or rotational velocity imposed (non zero) displacements, temperature (for thermal strain). A static analysis can be either linear or non linear. In our present work we are going to consider linear statistic analysis.

The procedure for static analysis consists of these main steps:

- Building the model
- Obtaining the solution.
- Reviewing the results.

## OBTAIN THE SOLUTION:

In this step we define the analysis type and options, apply loads and initial conditions the finite element solution. This involves three phases:

- Pre processor phase.
- Solution phase.
- Post processor phase.

## PRE-PROCESSOR

Pre processor has been developed so that the same program is available on micro, mini, super mini and main frame computer system to other. Pre processor is an interactive model builder to prepare FE (finite element) model and input data. The solution phase utilizes the input data developed by the pre processor, and dies the solution according to the problem definition. It creates input files to the temperature, etc., on the screen in the form of contours. The following section describes various capabilities and features of the pre-processor.

## MESH GENERATION

In finite element analysis the basic concept is to analyze the structure, which is an assemblage of discrete pieces called elements, which are connected together at a finite number of points called nodes. Loading boundary conditions are applied to these elements and nodes. A network of these elements is known as mesh.

## MATERIAL DEFINITION

All elements are defined by nodes, which have only their location defined. In the case of plate and shell elements, there is no indication of thickness. The thickness can be given as element property. Property tables for a particular property set

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

1-D have to be input. Different types of elements have different properties for e.g.

- Beams: cross-sectional area, moment of inertia etc.
- Shells: thickness;
- Springs: stiffness;
- Solids: none

The user also needs to define material properties of the elements. For linear static analysis, modulus of elasticity and Poisson ratio need to be provided. For heat transfer coefficient of thermal expansion, densities etc. are required. They can be given elements by the material property set 1D.

## SOLUTION

The solution phase deals with the solution of the problem according to the problem definitions. All the tedious work of formulating and assembling of matrices are done by the computer, and finally displacements and stress values are given as output. Some of the capabilities ANSYS are linear static analysis, transient dynamic analysis etc.

## POST PROCESSOR

It is a powerful user-friendly post processing program. Using interactive color graphics; it has extensive plotting features for displaying results obtained from the finite element analysis. One picture of analysis results (i.e., results in a visual form) can often reveal in seconds what would take an engineer hour to asses from a numerical output, saying a tabular form. The engineer may also see important aspects of the results that could be easily missed in a stack of numerical data. Employing state of the art image enhancement techniques, facilities viewing of Contours of stresses, displacements, temperatures etc.,

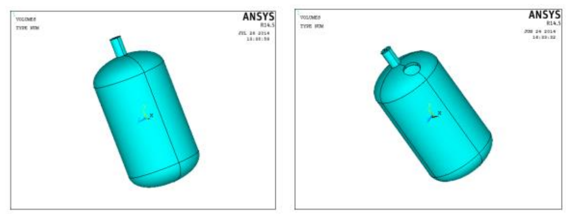
1) Deform geometric plots
2) Animated deformed shapes
3) Time history plots
4) Solid sectioning
5) Hidden line plot
6) Light source shaded plot .,

The entire range of post processing option are different types of analysis can be accessed through the command / menu mode there by giving the user added flexibility and convenience.

## CYLINDRICAL PRESSUREVESSELWITH AND WITH OUT HOLE

**Import iges Model from Pro/Engineer**

**STEEL**



Enter units in command prompt: /units, si, mm, kg, sec, k

**Preferences:** structural

## PREPROCESSOR:

**Main Menu> Preprocessor> Element Type> Add/Edit/ Delete**

**ADD Element Type**: solid 20 nodes 95

**Material Properties:**

Structural Properties: **(STEEL)**

Density – 0.00000785mm$^3$

Young's Modulus – 20900Mpa

Poisson's ratio - 0.284

**Generate Mesh**

**Main Menu> Preprocessor> Meshing> Mesh Tool >Smart size set to7**

[Mesh]

[Pick All]

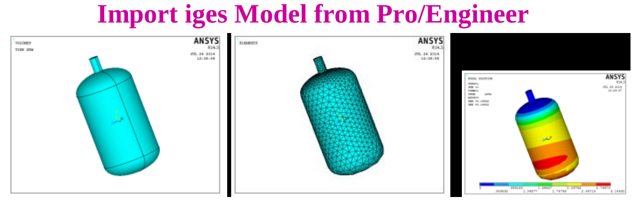
[Close]

Warning.

**Meshed Model**

## KEVLAR

**Import iges Model from Pro/Engineer**



Enter units in command prompt: /units, si, mm, kg, sec, k

**Preferences:** structural

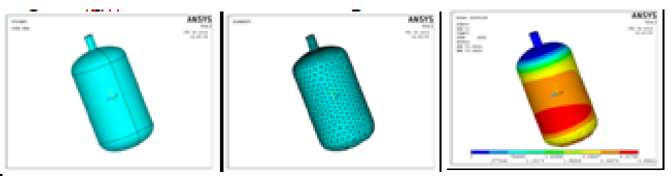
## PREPROCESSOR:

**Main Menu> Preprocessor> Element Type> Add/Edit/ Delete**

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

**ADD Element Type**: solid 20 nodes 95
**Material Properties**
Structural Properties: **(KEVLAR)**
Density – 0.00000144Kg/mm$^3$
Young's Modulus – 112000Mpa
Poisson's ratio - 0.36
**Generate Mesh Main Menu> Preprocessor> Meshing> Mesh Tool >Smart size set to7**
[Mesh]
[Pick All]
[Close] Warning.
**Meshed Model**

## S2 GLASS



## Import iges Model from Pro/Engineer

Enter units in command prompt: /units, si, mm, kg, sec, k
**Preferences:** structural
**PREPROCESSOR:**
**Main Menu> Preprocessor> Element Type> Add/Edit/ Delete**
**ADD Element Type**: solid 20 nodes 95

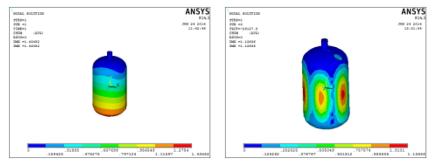
## Material Properties:

Material modals:
Structural Properties: **(S2glass)**
Density – 0.00000246Kg/mm$^3$
Young's Modulus – 86900Mpa
Poisson's ratio - 0.23
**Generate Mesh**

## RESULTS TABLE WITH AND WITHOUT HOLE

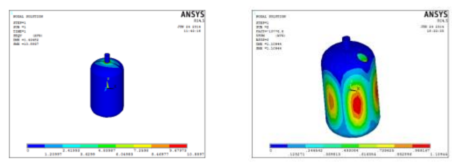
| | First Set | | Second Set | | Third Set | |
|---|---|---|---|---|---|---|
| | **DISP (mm)** | **FACTOR** | **DISP (mm)** | **FACTOR** | **DISP (mm)** | **FACTOR** |
| **Steel** | 1.13636 | 26127.6 | 1.03838 | 14174.5 | 1.1126 | 18729.7 |
| **S2 glass** | 1.04198 | 33784.4 | 1.10944 | 10776.6 | 1.18223 | 14924.3 |
| **Kevlar** | 1.02738 | 26142.2 | 1.11873 | 10771.2 | 1.12145 | 14913.9 |

## DISPLACEMENT



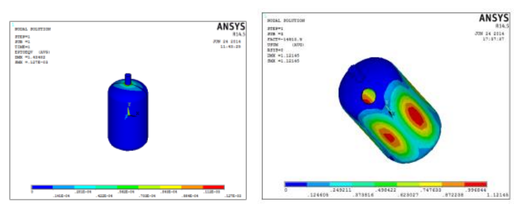
The minimum and maximum displacement is 1.43482mm and minimum displacement is 0.159425mm

## STRESS



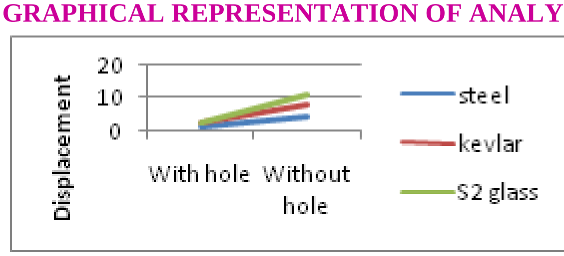
The minimum and maximum stress is in the range of 9.67973N/mm$^2$ to 10.8897N/mm$^2$.

## STRAIN

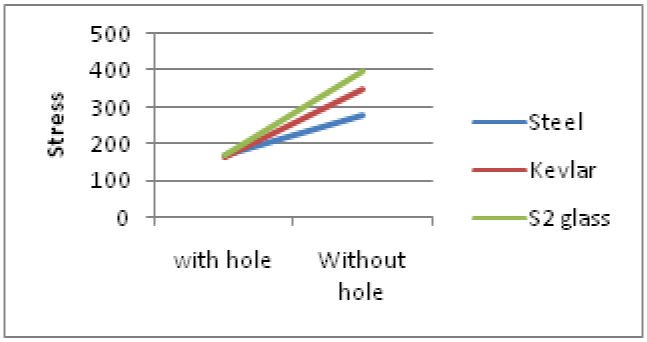


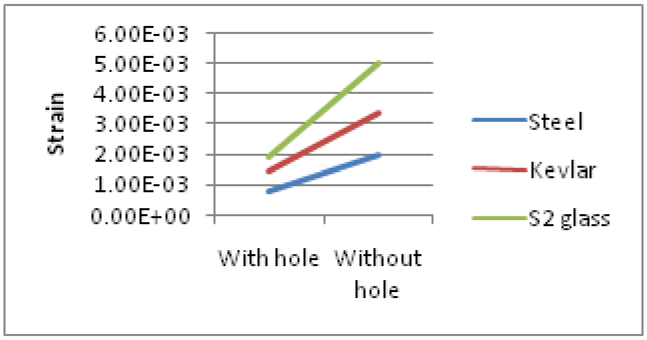
The minimum and maximum strain is 0.127e$^{-3}$ and minimum strain is 0.141e$^{-4}$

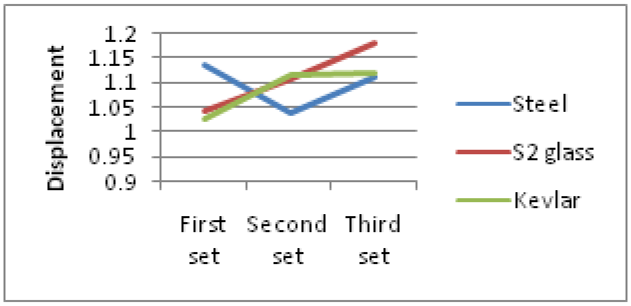
## GRAPHICAL REPRESENTATION OF ANALYSIS



ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal




## RESULTS TABLE FOR BUCKLING ANALYSIS

| | First Set | | Second Set | | Third Set | |
|---|---|---|---|---|---|---|
| | **DISP (mm)** | **FACTOR** | **DISP (mm)** | **FACTOR** | **DISP (mm)** | **FACTOR** |
| **Steel** | 1.13636 | 26127.6 | 1.03838 | 14174.5 | 1.1126 | 18729.7 |
| **S2 glass** | 1.04198 | 33784.4 | 1.10944 | 10776.6 | 1.18223 | 14924.3 |
| **Kevlar** | 1.02738 | 26142.2 | 1.11873 | 10771.2 | 1.12145 | 14913.9 |

## CONCLUSION

In this thesis, static and buckling analyses are to be done for determining the stress and deformation fields in a thin laminated-composite cylindrical shell with an elliptical head without cutout and with cutout are presented. Modeling is done in Pro/Engineer. The materials used are Steel, S2 Glass Epoxy and Kevlar. By replacing the materials with composites, the weight of the vessel decreases. Analysis is in Ansys. The analysis is done using solid element and shell element. 5 and 11 layers are used as layer stacking for composite materials S2 Glass and Kevlar. By observing the structural analysis results, the stress values are decreased for vessel with hole than the vessel without hole and the stresses and displacements are within the range of permissible values for every material. The stress values are almost reduced by 40% when steel is used, almost by 52.66% when Kevlar is used and almost by 58.19% when S2 Glass is used.

Structural analysis is also done using shell element for composite materials for 5 layers and 11 layers. By observing the results, the stress values are less for Kevlar than S2 Glass Epoxy. The stresses are reduced by 31.75% when 5 layers are taken and by 32.45% when 11layers are taken.
By using steel the failure chances are more than the composite materials. The factor is reduced by 20.31% for S2 Glass and by 20.37% for Kevlar. By comparing Kevlar and S2 Glass, the better material is Kevlar.

## REFERENCES

1) Stress analysis of composite cylindrical shells with an elliptical cutout by E. Oterkus, E. Madenci2) Buckling analysis of laminated composite plates using an efficient C FE model by S. K. Singh, A. Chakrabarti
3) Buckling Studies on Laminated Composite Skew Plates by C.V. Srinivasa, Y.J. Suresh,W.P. Prema Kumar
4) Elastic buckling analysis of laminated composite plates with through-the-width delamination using EAS solid element by Dae Yong Park Ph. D., Suk Yoon Chang, Sung Soon Yhim
5) Buckling Analysis of Woven Glass Epoxy Laminated Composite Plate by M Mohan Kumar, Colins V Jacob, Lakshminarayana N, Puneeth BM, M Nagabhushan