International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 04 Issue: 08 | Aug -2017 www.irjet.net p-ISSN: 2395-0072

DESIGN AND FE ANALYSIS OF COMPOSITE GRID STRUCTURE FOR SKIN STIFFENING APPLICATIONS

NITISH GHADI[1], PROF.ANAND MATTIKALLI[2], PROF. ANANT GHADI[3].

[1]Post Graduate Student, Department of Mechanical Engineering, MMEC Belgaum, Karnataka, India
[2,3]Professors, Department of Mechanical Engineering, MMEC Belgaum, Karnataka, India

***

Abstract - *A grid structures are the shell like structures, which supports the skin of any structure. When made up with composite materials Grid structures find very good application in aerospace field. The properties of the skin can be uniformly distributed, thickness of the skin can be reduced which intern reduces the total weight of the structure. The present skin stiffened structures are having thick skin and it is contributing more weight and it is taking some part of the load. By using grid structures we can reduce the skin thickness and load shared by the skin can be minimized which gives the damage tolerant design concept for aerospace structures and it also lays foundation for the futuristic adaptive structures. Therefore presently composite grid structure analysis is conducted to know the effectiveness of the various grid structures in skin stiffening applications*

Key Words: Intrinsically stiff, flexural stiffness, Delimitation, Reinforced, Contamination

1. INTRODUCTION

1.1 Grid structures

Grid structures are the shell like structures, which supports the skin of any structure.

1.1.1 Types of Grid Structures

There are several types of standard grid structures. Important among them are as follows:

- Grid structures with ribs running in four directions are referred to as quadri- directional grids.
- Grid structures with ribs that are in three directions are referred to as tri-directional grids. An Iso-grid is a special case of tri-directional grid structure in which the ribs form an array of equilateral triangles.
- Grid structures with ribs drawn in only two directions are referred to as angle grids and if the two directions are orthogonal then this structure is referred to as ortho-grid.

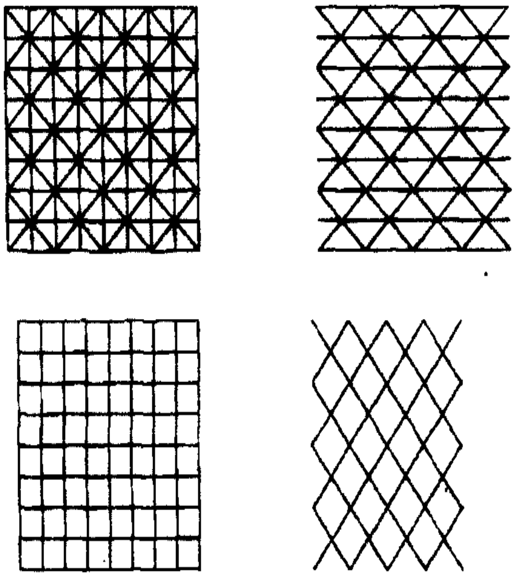

Fig. 1.1 Grid structures

1.1.2 Advantages of grid structures

- As all ribs are made of continuous and unidirectional fibers, they are intrinsically stiff, strong and tough.
- Composite grid structures are damage tolerant. Ribs are unidirectional so that they do not delaminate and crack is not likely to propagate across the spacing between ribs or the interface between ribs and skins.
- There exists a potential for completely automating the fabrication process. So fabrication of the composite grid structures is easy and less time consuming.
- Grid structures are open structures and thus easy to inspect or repair.
- Grid structures are generally added to flat or curved skin panels to
    - ➢ Increase out – of – plane flexural stiffness.
    - ➢ Resist flexural loads with additional cross sectional area to withstand axial loads.
    - ➢ Act in addition as fins to dissipate heat and moisture.

1.1.3 Application of grid structures

Grid structures are extensively used in aerospace, automobile and in civil structural applications. The grid structures consist of inherent resistance to impact damage, delimitation and crack propagation. Grid structure behavior study is inescapable, before implementation. Since the aerospace structures are subjected to combined loading situations, a proper study must be carried out for the grid structure model but not under single load case, but as multi-directional surface in failure space, which is termed as failure envelope.

1.2 Composite materials

Composite materials are the materials that are made from two or more constituent materials with notably different physical or chemical properties, that when merged, produce a material with characteristics that is different from the single components. The individual components continue to exist separately and distinctly within the finished structure. The new material may be preferred for several reasons: commonly include materials which are stronger, lighter or inexpensive when compared to traditional materials.

1.2.1 Importance of composite materials

The properties of composite materials cannot be achieved by the constituent materials alone. It is observed that composites are becoming extremely important as it can help to improve the quality of our life. Composites are utilized in flight vehicles, automobiles, boats, pipelines, buildings, roads, bridges, and dozens of other products. Researchers are trying to find out ways to improve qualities of composites which would mean they may be strong, lightweight, long-lived, and inexpensive to produce.

1.2.2 Different types of composites

- Natural composites

Producing composites is an attempt to replicate nature. Example wood is a composite of cellulose fibers that are cemented together with lignin.

- Man-made composites

They are made by adding reinforcing fibers into polymer matrix, metal matrix or ceramic matrix respectively.

    - Polymer matrix composites (PMCs)
    - Metal matrix composites (MMCs)
    - Ceramic matrix composites (CMCs)
    - Carbon - Carbon Composite (CCC s)
    - Intermetallic Matrix Composites (IMCs)

Polymer matrix composites (PMCs)

The most common composites are the Polymer Matrix Composites (PMCs) that are also known as Fibre Reinforced Polymers (or Plastics) FRP. These substances use a polymer-based resin as the matrix, and also a variety of fibres such as glass, carbon and aramid as the reinforcement. These matrix materials are thermosetting thermoplastic, rigid rod plastic, elastomer, and thermo elastic plastic polymers. The reinforcing fibres are continuous or chopped. In summary, polymer composites processing includes contracting of polymer and fibres, shaping, controlled heating and / or reactions, and cooling process. This technique of polymer composites has expanded its scope to larger extent in aerospace arid military applications.

Metal matrix composites (MMCs)

Metal matrix composites can be acquired using either a primary liquid phase approach like squeeze casting/infiltration or spray deposition, or a primary solid state processing such as powder techniques and foil diffusion. Common MMCs are aluminium based MMCs, fibre reinforced titanium alloys, and magnesium alloy-SiC particulate MMC. The aluminium-based materials are the most popular due to their cost and easy fabrication.

Ceramic matrix composites (CMCs)

Ceramic Matrix Composites (CMCs) are utilized in very high temperature environments. The definition of the ceramic matrix can be broader. It includes inorganic silica-based glass, crystalline ceramics, glass-ceramics, intermetallic and carbon. All of these have an unique unifying thread and they are fairly high temperature structural materials. In CMCs, the ceramic matrix covers a wide variety of inorganic materials, which are usually non-metallic and are processed in high temperatures. Common ceramic matrix materials include various glasses, glass ceramics and ceramics, such as carbon, silicon carbide, silicon nitride, aluminised and oxides. These reinforcements can include carbides, borides and oxides.

Carbon - carbon composite (CCC s)

The development of carbon-carbon materials began in early 1958 and was nurtured under the US Air Force space plan program DynaSoar, and NASA 3 Apollo projects Carbon-carbon materials are a generic class of composites like graphite /epoxy family of polymer matrix composites. They can be created in a wide variety of forms. From one dimensional to non-dimensional using unidirectional tow tapes or woven cloth. Because of their multiple formations, their mechanical properties can be readily tailored. Carbon materials have high strength and potential to stay stiff as well as high thermal and chemical stability in inert environment .They must however, be protected with coatings and /or surface sealants when used in an oxidizing environment.

Intermetallic Matrix Composites (IMCs)

Several problems limit the development of inter-metallic-matrix composites (IMCs) in majority including chemical incompatibility and CTE mismatch between potential reinforcement of fibres and matrix materials, poor-to-low-temperature ductility, and marginally high-temperature oxidation resistance of intermetallic materials. Composite fabrication and joining processes do not result in excessive fibre/matrix reaction or matrix contamination. The beginning phase of the IMC program involves investigating available fibre compositions (SiC and Al203) in aluminised of iron, titanium, nickel, and niobium. These aluminise are Ti3 AI and Fe3C for applications to 1000°C and NiAI and Nb-alloy/aluminised for higher temperature applications. The studies of alloying these materials are aimed at increasing toughness, ductility, and oxidation resistance, and promoting long time stability with the candidate fibre materials.
Candidate matrices will be evaluated using tensile, compression, fatigue, creep, and oxidation tests. Measuring appropriate thermal and physical properties is another planned task. Powder-cloth fabrication processes have been developed to produce IMC materials, and alternative processing procedures, such as thermal spraying, are being Ti3Al + Nb material, based on tensile, thermal-cycle, and strain-controlled fatigue studies for temperatures up to 815°C. Studied Encouraging results have been obtained on SiC---reinforced.

1.3 Objectives of the present work

In the present work orthogrid and isogrid concepts will be used for skin stiffening applications. In the first step of our project a rectangular panel or plate is designed with Orthogrid and isogrid skin stiffener for some desired load conditions with same skin thickness. The designed plates are analyzed for deflection and stress using finite element analysis (FEA) software such as Ansys. In the second step the skin thickness of ortho-grids is changed to get the same weight of both the plates. Conclusions will be derived related to the deflection, stress distribution and stiffness to weight ratio of the designed structures.

2. ANALYSIS OF COMPOSITE PLATE

2.1 Introduction to Finite Element Analysis (FEA)

The Finite Element Method (FEM) is a numerical procedure for getting approximate answers to many of the problems encountered in Engineering. In FEM the complete region is discretized in to a number of geometric shapes, which are called as elements. The governing relationships and the properties are assumed over there elements and expressed mathematically at specific points which are called nodes. An assembly procedure is used to link individual elements and the effects of loading and boundary conditions applied over these elements to get a set of relations either linear or nonlinear solutions to these equations would provide approximate solution of that particular system.

The FEM originated in the design of aircrafts as a method of stress analysis. It started as an extension of matrix method of structural analysis. Today this FEM method is used not only for the analysis in solid mechanics, but even in the analysis of heat transfer, liquid flow, electric and magnetic field and civil engineering. This method is extensively used for the analysis of beams, space frames, plates, sheets etc. in aforementioned fields. This method is also used for critical and complicated analysis like the analysis and design of ships, aircrafts, space crafts, heat engines etc.

Today the developments in main frame computer and availability of powerful microcomputer have brought this method within research of students and engineers working in small industries.

2.1.1 Steps involved in FEA

The first step in finite element analysis is to model the given system. In this two distinct approaches to model physical systems, the discrete lumped approach and the continuum approach. In the former approach the factual system is initially idealized as an assembly of the elements and the equilibrium equations is given for each element and the result is used to set of equations is solved to yield the solutions. In the later approach these actual system is treated as one continuum and one or more equilibrium equations are written which are then solved for the system response.

The given continuum after being divided into finite elements is called finite element model. A typical finite element model is comprised of nodes, elements material properties, degrees of freedom, boundary conditions, externally applied loads and analysis type. Engineers and designers must carefully create the model to ensure the relevance of the results of the corresponding FEA. These results depend solely on the model.

In practice, practitioners must decide on the mesh layout, i.e. the number of nodes and elements. Zones of expected abrupt changes in the unknown variables (such as stress concentration around holes) require a denser number of nodes and elements then zone where gradual changes occur, this known as gradation. After the mesh layout is chosen practitioners must choose the analysis (static/dynamic, linear/nonlinear, beam, theory plane stress, three dimensional etc.) the type (deflection, rotation, temperature, flux etc.) and number of degrees of freedom at each node, the boundary conditions, relevant information (type, number of nodes per element etc.), material properties and dumping external loads at the nodes.

There are many alternative to solve linear and nonlinear boundary and initial value problem, ranging from completely numerical. Exact solutions are usually available for a few solutions usually with simple domains (e.g. regular domain). This solution can be obtained by direct integration of the differential equations. This can be achieved by techniques such as separation of the variables at Fourier and Laplace transformations to name only a few. Approximations are usually sought of approximate technique exist and include perturbations, power series probability scheme, the method of weighted residuals the finite difference method, the Raleigh-Ritz method and the finite element method.

Once the finite element model is defined by choosing all of the above parameter of the corresponding mesh, it must be input to the code that performs the FEA. A review of the most exciting FEA commercial codes reveals that require there user to provide the data of the finite model in a data of the finite model in a data file with specification detailed format of a certain data file, such a file usually has five major sections: control, nodal, element, material and loading sections. While later four sections are self-explanatory, the control section includes information such as the problem description (heading), the total number nodes, the type of analysis etc.

These codes have the six modules.

- **Node module:** Node generates degree of freedom, and nodal co-ordinates.
- **Boundary condition module:** It stores material properties in the library of material.
- **Element module:** It various element types in an element library, evaluates element matrices and assemblies these matrices.
- **Loading module:** It applies the externally applied loads to the proper nodes and elements.
- **Solution module:** This module solves the system of equation that results from the assembly process of the element module for the unknown modules.

FEA codes consist of pre-processors and post-processor. Pre-processors and post-processor are classified as much relative to FEA process. Pre-processing precedes this process and help the user to automate and/or facilitate the input required by the first modules described in the results in various graphical forms to get better understand and interpret them.

The output from FEA codes is primarily in numerical form. It usually consists of the nodal values of the unknown variables and derivatives. For e.g.: In solid mechanics problems the output is the nodal temperatures and element stress, in heat transfer problems the output is the nodal temperatures and element heat fluxes. Graphical outputs are usually more informative in providing trends of continuum behavior. Thus curves and contours of the field variable can be plotted and displayed. Also deformed shapes can be displayed superposed on unreformed shapes.

2.1.2 Advantages of the FEA

- FEA method Can readily handle complex geometry:
- FEA method can handle complex analysis types such as: Vibration, Transients, Nonlinear, Heat transfer, Fluids etc.
- FEA method Can handle complex loading: Node based loading (point load), Element based loading (pressure, thermal, inertial forces), Time or frequency dependent loading.
- FEA method can handle complex restraints: Indeterminate structures can be analyzed.
- FEA method can handle bodies comprised of non-homogeneous materials: Every element in the model could be assigned a different set of material properties.
- FEA method Can handle bodies comprised of non-isotropic materials: Orthotropic, Anisotropic
- FEA method Special material effects are handled: Temperature dependent property, Plasticity, Creep, Swelling.

2.2 Composite plate analysis

In the present work the composite plate with the following dimensions is considered for the FEA modeling and analysis.

Dimensions of the plate: width = 500 mm, length = 700 mm, skin thickness = 3 mm

The analysis has been carried out with two grid structures: ortho-grid and iso-grid and the results are compared in order to check that which grid structure enhances the stiffness properties of the skin.

Finite element modeling

As it is a composite material for plate modeling a structural element is used that is shell linear layer 99. In this element the thickness is built by using the number of layers of laminates. A 5 layered symmetric shell plates with bidirectional i.e. 0-90-0-90-0 laminates with each layer of thickness 0.5 mm is used to build the skin thickness of 3 mm. Carbon- Epoxy composite material is considered in the present work. Carbon is isotropic material and it has Young's Modulus of 39500 MPa and Poisson's ratio 0.245.

Then in the modeling part a plate of length 700 mm and width 500 mm is created. On that skin ortho grids and iso grids are generated in which ribs are placed orthogonal i.e. 90^0 and 60^0 to each other respectively. Then the area of skin is divided with reference to ribs and mapped meshing is done for each area of ribs and skin.

Material properties

Material = Carbon/epoxy composite material

Transverse tensile strength of carbon/epoxy = 600 MPa

Longitudinal compressive strength of carbon/epoxy = 570 MPa

Density = 1.6 X 10^{-6} Kg / mm^3

(Reference - http://www.azom.com/article.aspx)

2.2.1 Skin with ortho-grid

For the present analysis ortho-grids structure rib thickness and rib depth are taken as 3 mm and 25 mm respectively. Skin thickness is taken as 3 mm.

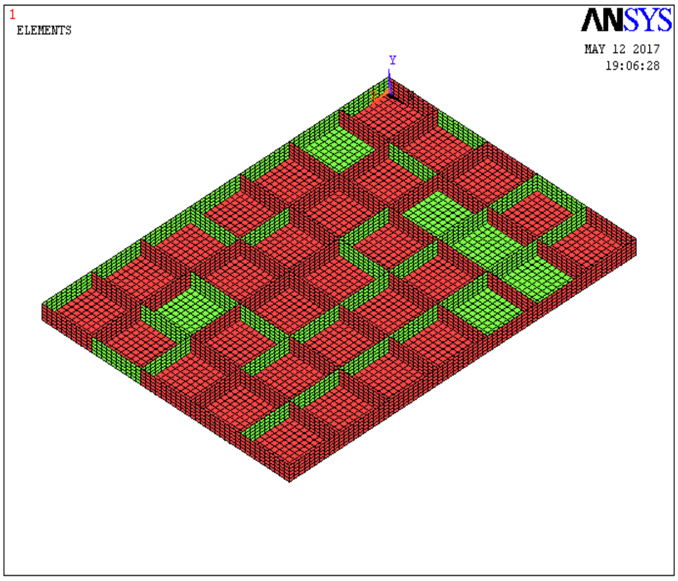

Fig.2.1 Meshed model of the composite plate with ortho-grid

Load Case 1: Two ends fixed with transverse loading boundary conditions

Two opposite ends that are of length 500 mm are fixed and a transverse load of 100 KN is applied on the skin.

Now the uniform pressure that is to be applied on the skin is given by Eq. (2.1).

$$P = F/A \qquad (2.1)$$

$$P = 100\times10^{3}/(700\times500)$$

$$P = 0.285\,\mathrm{MPa}$$

This magnitude of pressure is applied on the skin.

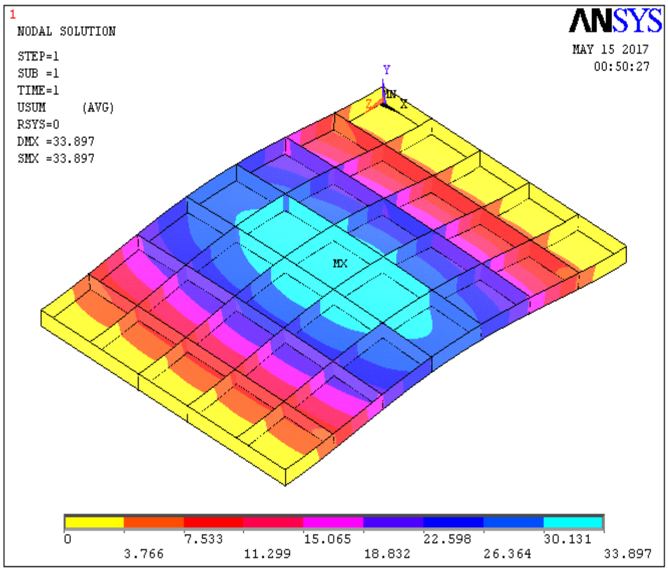

Fig. 2.2 Two ends fixed with transverse load (deflection)

Result

After the analysis it is found that the maximum deflection is at the center of the plate and its magnitude is 33.897 mm.

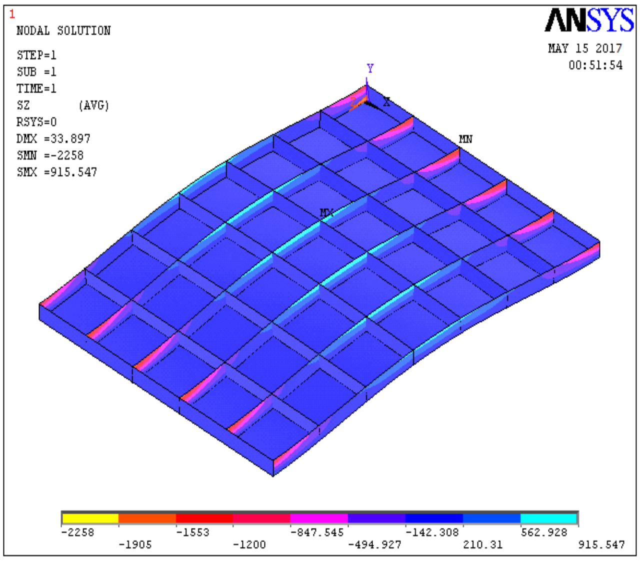

Fig. 2.3 Two ends fixed with transverse load (Stress)

Result

After the analysis a maximum stress of 915.547 N/mm^2 is observed at the center of the plate. Near the fixed ends the stress development is minimum.

Load Case 2: One ends fixed axial loading boundary conditions

One end of length 500mm is fixed and axial load of 1000 N is applied on the ribs on opposite side.

$$P = 1000/(500*25)$$

$$P = 0.08\,\text{MPa}$$

This magnitude of pressure is applied on the skin

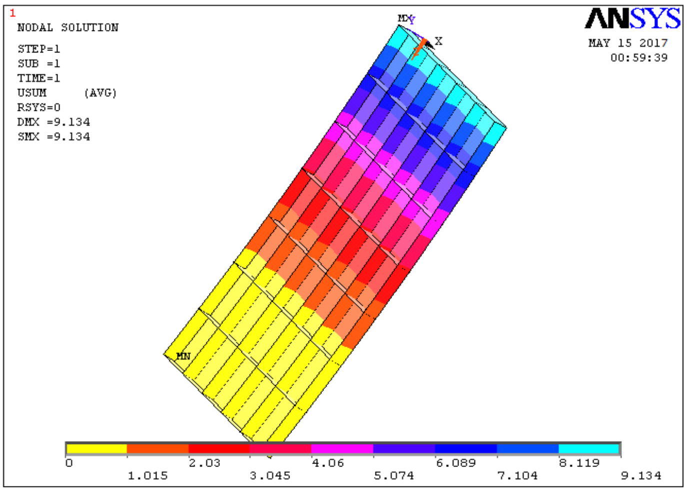

Fig. 2.4 one ends fixed axial loading (deflection)

Result

After the analysis a maximum deflection of 9.134 mm is observed at the free end of the plate.

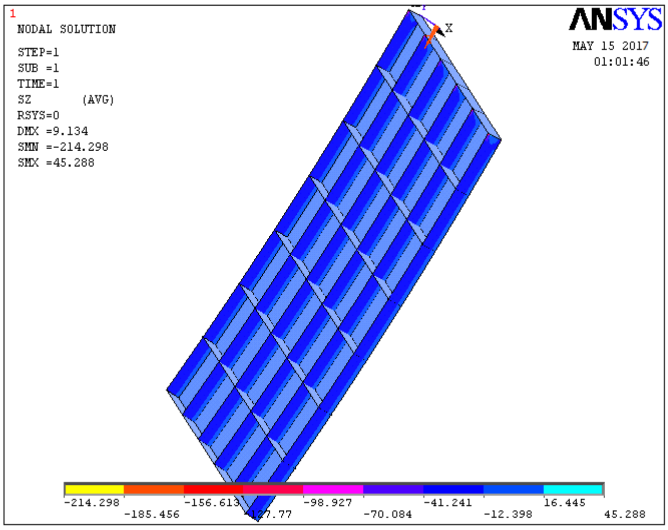

Fig. 2.5 One end fixed axial load (Stress)

Result

After the analysis a compressive stress of 214. 298 N / mm^2 is induced at the fixed end of the plate and at the lower side of the rib. Towards free end the stress is minimum and its magnitude is 45.288 N / mm^2.

2.2.2 Skin with Iso-Grid

The rib dimensions for the ortho-grids are: thickness = 3 mm, depth = 25 mm and the skin thickness is 3 mm.

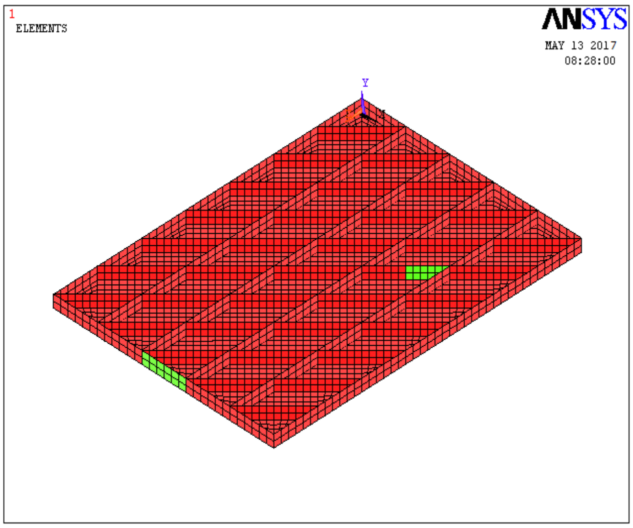

Fig.2.6 Meshed model of the composite plate with iso-grid

Load Case 1: Two ends fixed with transverse loading boundary conditions

Two opposite ends that are of length 500 mm are fixed and transverse load of 100 KN is applied on the skin.

$$P = 100\times10^{3}/(700\times500)$$

$$P = 0.285\,\text{MPa}$$

This magnitude of pressure is applied on the skin.

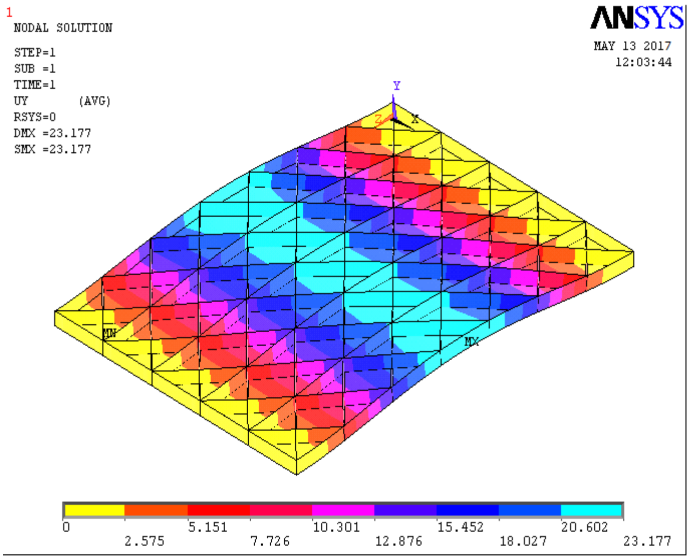

Fig. 2.7 Two ends fixed with transverse load (deflection)

Result

After the analysis it is found that the maximum deflection is at the center of the plate and its magnitude is 23.177 mm.

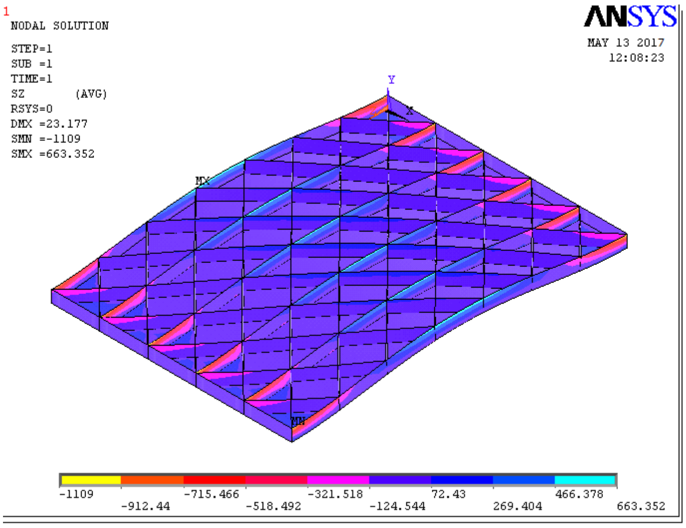

Fig. 2.8 Two ends fixed with transverse load (Stress)

Result

After the analysis a maximum stress of 663.352 N/mm^2 is observed at the center of the plate. Near the fixed ends the stress development is minimum.

Load Case 2: One ends fixed axial loading boundary conditions

One end of length 500mm is fixed. Axial load of 1000 N is applied on the ribs on opposite side.

$$P = 1000/(500*25)$$

$$P = 0.08\,\text{MPa}$$

This amount of pressure is applied on the skin.

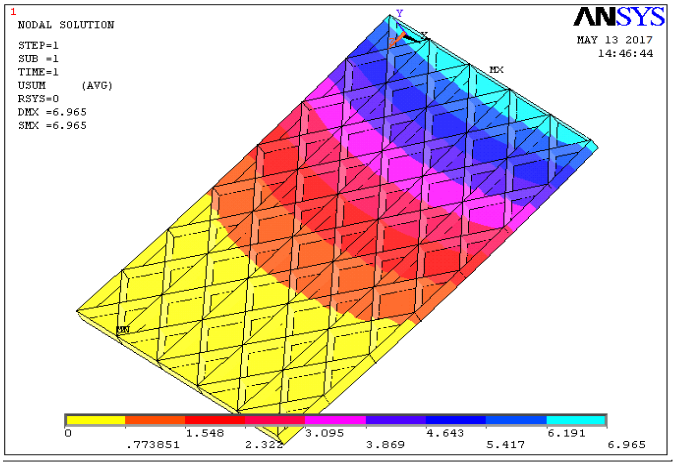

Fig. 2.9 one ends fixed axial loading (deflection)

Result

After the analysis a maximum deflection of 6.965 mm is observed at the free end of the plate.

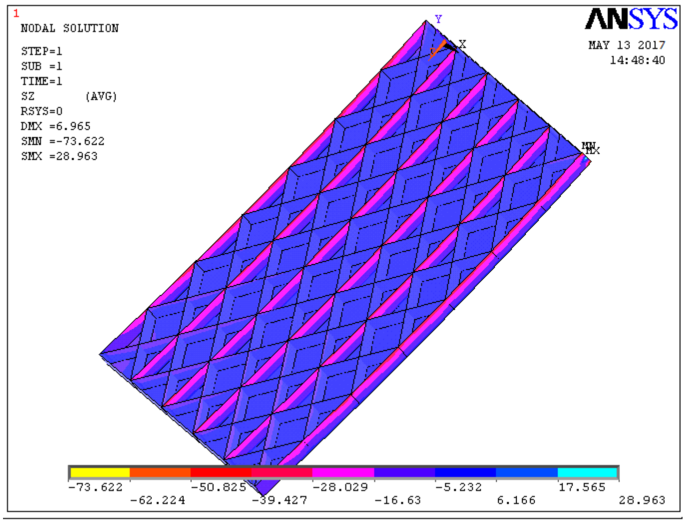

Fig. 2.10 One end fixed axial load (Stress)

Result

After the analysis a compressive stress of 73.622 N / mm^2 is induced at the fixed end of the plate and at the top side of the rib. Towards free end the stress is minimum and its magnitude is 28.963 N / mm^2.

Reduced thickness of Iso-grid plate

Now for the same amount of load the skin and rib dimensions for the Iso-grids structure are reduced to 2.22 mm and the analysis is carried out.

Load Case 1: Two ends fixed with transverse loading boundary conditions

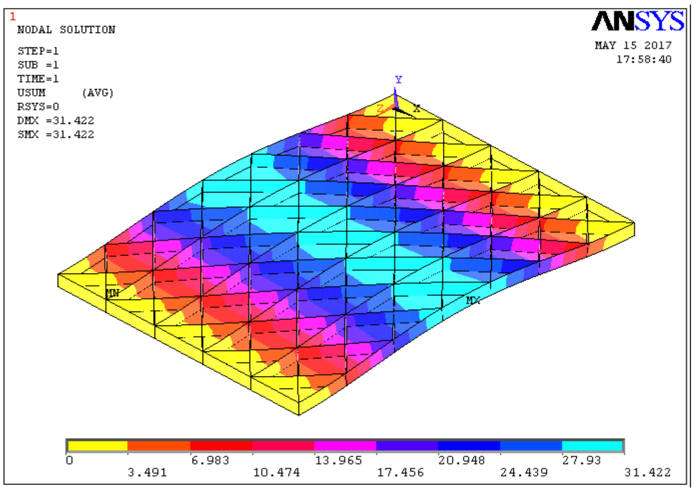

Fig. 2.11 Two ends fixed with transverse load (deflection)

Result

After the analysis it is found that the maximum deflection is at the center of the plate and its magnitude is 31.422 mm.

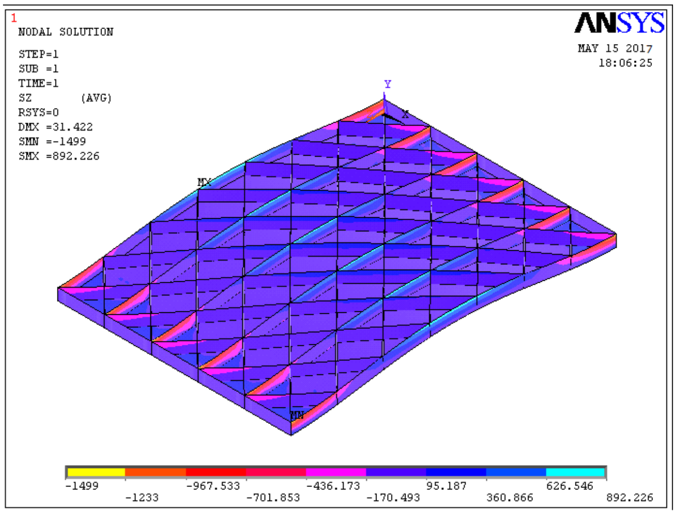

Fig. 2.12 Two ends fixed with transverse load (Stress)

Result

After the analysis a maximum stress of 892.226 N/mm^2 is observed at the center of the plate. Near the fixed ends the stress development is minimum.

Load Case 2: One ends fixed axial loading boundary conditions

One end of length 500mm is fixed and axial load of 1000 N is applied on the ribs on opposite side.

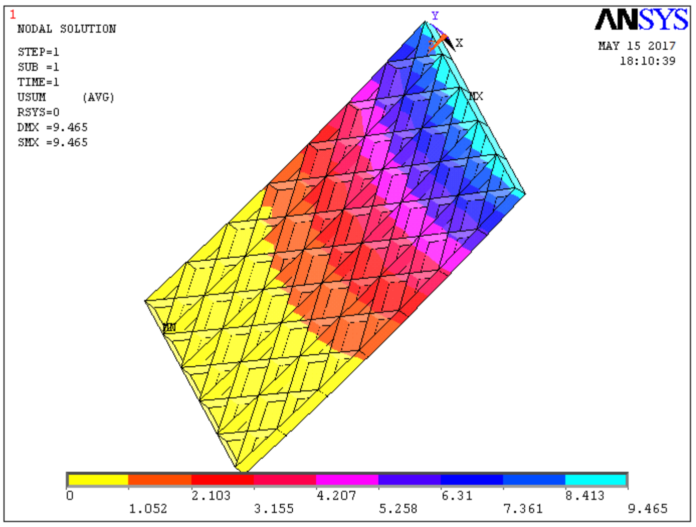

Fig. 2.13 one ends fixed axial loading (deflection)

Result

After the analysis a maximum deflection of 9.465 mm is observed at the free end of the plate.

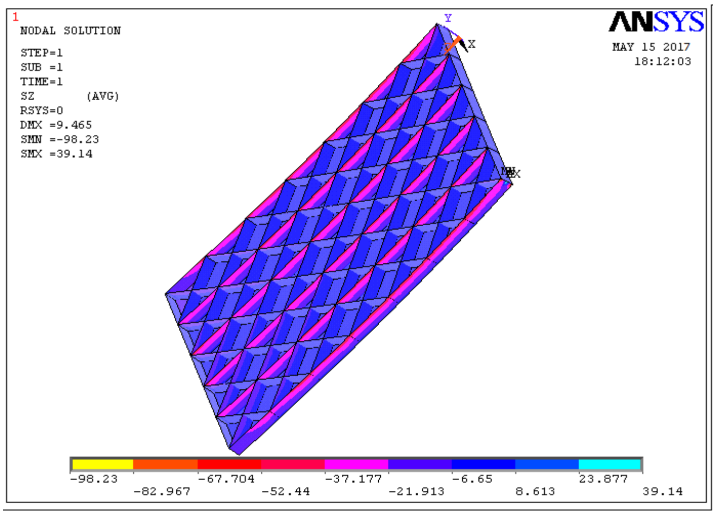

Fig. 2.14 One end fixed axial load (Stress)

Result

After the analysis a compressive stress of 98.23 N / mm^2 is induced at the fixed end of the plate and at the top side of the rib. Towards free end the stress is minimum and its magnitude is 39.14 N / mm^2.

2.2.3 Analytical calculation

Stiffness (k)

For both the plates Transverse load = 100 KN and Axial load = 1000N

For Transverse load

Ortho grid,

Stiffness = Force / Displacement

k= 100e3 / 33.897

k = 2950.11 N/mm

Iso grid,

Stiffness = Force / Displacement

k= 100e3 / 23.177

k =4314.62 N/mm

For Axial load

Ortho grid,

Stiffness = Force / Displacement

k= 1000 / 9.134

k = 109.48 N/mm

Iso grid,

Stiffness = Force / Displacement

k= 1000 / 6.96

k = 143.678 N/mm

Weight (W)

Weight = Mass * Acceleration of Gravity

But Mass = Density * Volume

For Ortho Grid

Volume V = Volume of Plate + Volume of Ribs

V= (700*500*3) + ((700*3*25)*6 + (500*3*25)*8)

V = 1612500 mm^3

Mass M = 1.6*e-6 * 1612500

M = 2.58 Kg

W = 2.58 * 9.81

W = 25.30 N

For Iso-Grid

Volume V = Volume of Plate + Volume of Ribs

V= (700*500*2.22) + [(700*2.22*25)*6 + (500*2.22*25)*2 +(70*2.22*25)*140

V = 1609500 mm3

Mass M = 1.6*e-6 * 1609500

M = 2.57 Kg

W = 2.57 * 9.81

W = 25.27 N

Reduced thickness of Iso-grid structure Stiffness (k)

For both the plates Transverse load = 100 KN and Axial load = 1000N

For Transverse load

Iso grid,

Stiffness = Force / Displacement

k= 100e3 / 31.42

k = 3182.48 N/mm

For Axial load

Iso grid,

Stiffness = Force / Displacement

k= 1000 / 9.46

k = 105.70 N/mm

Weight (W)

Weight = Mass * Acceleration of Gravity

Mass = Density * Volume

Volume V = Volume of Plate + Volume of Ribs

= (700*500*2.23) + (700*2.23*25)*6 + (500*2.23*25)*2 + (70*2.23*25)*140

V = 1616750 mm^3

Mass M = 1.6*e-6 * 1616750

M = 2.58 kg

W = 2.58 * 9.81

W = 25.31 N

3. CONCLUSIONS

In the present work the effect of orth-grid and iso-grid on the composite skin structure is studied through finite element analysis. Results derived from the analysis are tabulated below and the conclusions made from the analysis are also discussed.

Table 5.1 Analysis results for same skin thickness

Boundary Conditions	Ortho - grid structure	Iso - grid structure	% Error
1.Weight	25.30 N	34.14 N	34.94
2.Two end fixed Transverse load			
Deflection	33.897 mm	23.177 mm	46.25
Stress	915.547 N/mm^2	663.352 N/mm^2	38.01
Stiffness	2950.11 N/mm	4314.6 N/mm	46.25

3.One end fixed axial load			
Deflection	9.134 mm	6.96 mm	31.23
Stress	45.288 N/mm^2	28.963 N/mm^2	56.36
Stiffness	109.48 N/mm	143.678 N/mm	31.23

Table 5.2 Analysis results for reduced skin thickness of iso-grid

Boundary Conditions	**Ortho - grid structure**	**Iso - grid structure**	**% Error**
1.Weight	25.30 N	25.30 N	0.0
2.Two end fixed Transverse load			
Deflection	33.897 mm	31.422 mm	7.87
Stress	915.547 N/mm^2	892.226 N/mm^2	2.61
Stiffness	2950.11 N/mm	3182.48 N/mm	7.87
3.One end fixed axial load			
Deflection	9.134 mm	9.46 mm	3.56
Stress	45.288 N/mm^2	39.14 N/mm^2	15.70
Stiffness	109.48 N/mm	105.70 N/mm	3.57

From the above results it can be observed that for the same skin thickness under transverse loading deflection in the iso-grid structure is 37% less than the orth-grid. Structural stiffness is enhanced by 46.25% in iso-grid. Also the maximum stress induced in the iso-grid plate is 38% lower than the ortho-grid plate. For the axial loading the iso-grid structure provides 31.23% higher stiffness than the ortho-grid. However the weight of iso-grid structure is found to be 35% higher than the ortho-grid structure. Therefore to reduce the weight of the iso-grid structure the ribs and skin thickness is reduced by 26%. Now the weight of iso-grid plate is same as that of orth-grid plate. Analysis is carried out again to check the performance of the iso-grid plate with reduced weight. In this case also the iso-grids perform better than the ortho-grids in transverse load condition. However in axial loading case the stiffness of the iso-grid is reduced by 3.4 %. This is because the cross sectional area in the axial loading is more for ortho-grid compared to iso-grid. However this difference is minimum because the diagonal members of iso-grid structure have also contributed to support the axial load.

So finally it can be conclude that iso-grid structures are better than ortho-grid in skin stiffening applications.

ACKNOWLEDGEMENT

It gives us an immense pleasure to acknowledge all the people who helped us in successful completion of this project.

We take this opportunity to express our most sincere gratitude and profound regards to our beloved Principal *Prof. V.R.UDUPI* for providing all the facilities to do this Project.

Also we express our deep sense of gratitude and appreciation to our beloved H.O.D. *Prof. S. S.MASHYAL* for this enthusiastic inspiration in all phases of our Project.

With due respects we would like to express our sincere thanks to our Project Guide *Prof. A. C.MATTIKALLI* for his sterling efforts and timely guidance, patience in solving our doubts, which kept cropping up in due course of our project work needs special credits.

Our sincere thanks to all the people who have contributed to complete this Project successfully.

REFERENCES

1. Knou H., Nabavi S.M., Jam J.E., 2005. “Modeling and analysis of composite grid structure.”
2. Adam B., Higgins J., 2007. “Design and evaluation of reinforced advanced grid stiffened composite structure.”
3. Blom A. W., Russian M., Stickler P. B., 2010. “Bending test of variable stiffness fiber reinforced composite cylinder.”
4. Al-Mosawi A. I., Al-Maamori M. H., Wetwet Z.A., 2012. “Mechanical properties of composite material reinforcing by natural synthetic fibre.”
5. Arashamehr J., Rahimi G.H., Rasouli S. F., 2013. “Experimental and numerical study of a stiffened composite cylindrical cell with clamped free boundary conditions under transverse end loading.”
6. Mukhopadhyay M., 2009. “Mechanics of composite materials and Structures.”
7. Das J.B.K., 8th edition 2013. “Design of Machine Elements-1.”
8. Lingaiah K., 2006. “Design Data Handbook.”
9. Mikulas M. M., 1978. “Iso-grid Design Handbook.”
10. Swamy A.R.K., “Finite Element Method.”

BIOGRAPHIES

Mr.Nitish Ghadi post graduate student of the Dept. of Mechanical Engg, Maratha Mandal Engineering College Belgaum. Area of Interest
in machine design, Manufacturing Tech.

Mr.Anand Mattikali is a faculty member in the Dept. of Mechanical Engg, Maratha Mandal Engineering College, Belgaum. His areas of interest include machine design, Manufacturing Tech.

Mr. Anant J. Ghadi is a faculty member in the Dept. of Mechanical Engineering, Maratha Mandal Engineering College, Belgaum. His areas of interest include machine design, manufacturing technology and its applications to agriculture.