International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 2, February - 2014

Finite Element Modelling For Stress Analysis of a Rhombic Skew Plate Structure

[1.] Manjunath G. A, [2.] Athresh Kumar R, [3.] Arvind B, [4.] Punith R
[1.] Asst.Professor, Department of Mechanical Engineering, SJB Institute of Technology, Bangalore
[2,3,4.] Department of Mechanical Engineering, SJB Institute of Technology, Bangalore

Abstract— **Skew plate structures can be found frequently in modern construction in the form of reinforced slabs or plates, stiffened or fibre reinforced plastics super structures deck or skew grid of beams and girders. Such structures are widely used as floors in bridges, ship hulls, buildings and so on. Static stress analysis of rhombic skew plate subjected to uniform distributed load by using finite element method is investigated. A practical procedure for rhombic skew plate static analysis is performed to obtain the maximum stress induced in the skew plate. Two dimensional Finite element models of skew plate with 30^0,45^0 and 60^0 skew angles and analysis is performed considering uniform distributed load and simply supported conditions. The Model under suitable boundary conditions and loads is subjected to Stress Analysis. Submodeling is used to capture the variation of stresses at corner for both tapered skew plate and rhombic skew plate of 30^0 angles.**

Keywords— Finite Elements, Mechanics Of Solids, Skew Plate, Stresses, Stress Analysis.

I. Introduction

A. Background

Skew plate structures can be found frequently in modern construction in the form of reinforced slabs or plates, stiffened or fiber reinforced plastics super structures deck or skew grid of beams and girders. Such structures are widely used as floors in bridges, ship hulls, buildings and so on. However, the research into skew plate bending problems has not received as much attention compared with rectangular, circular and elliptical plate bending problems. There is a very strong singularity at the obtuse vertex along with increasing skew angle.

The analysis of the structural behaviour of skewed plates poses some difficulties due to the singularities which may develop at the corners especially for large degrees of skewness and/or for specific boundary conditions as discussed by Timoshenko and Woinowsky-Krieger. This singular behaviour, as described by Williams, explains the occurrence, at the obtuse corners of skewed plates, of a great number of cracks and/or excessive deformation typical of skewed plates. Despite the numerical difficulties in the analysis and the problems described, the use of skew plates in structures is increasing.

Skew plates are extensively used in various mechanical, civil and aero structures and they are mostly subjected to uniform pressure loading in its transverse faces. Specific application of isotropic skew plates includes aircraft wings and aircraft tail-fins. The edges of these plates are often so mounted that their boundaries can be assumed to be equivalent to various classical flexural boundary conditions. Static behaviour of the skew plates has significant contribution for its mechanical design, the present paper deals with the simulation of static behavior of skew plates under uniform pressure loading.

Skew plates are often used in modern structures in spite of the mathematical difficulties involved in their study. Swept wings of aero planes can be idealized by introducing substitute structures in the form of skew plates. Complex alignment problems in bridge design are often solved by use of skew plates due to functional, aesthetic or structural requirements. Various other applications of skew plates can be found in ship hulls, as well as parallelogram slabs in Buildings.

Practicing engineers may have to come across the problems of transverse bending of skew plates in this course of the design and construction of structures. A symmetric study and analysis of such problems for different angles of skew and support conditions may help them understand the behavior of such structures and prepare the prerequisites for design works. Published results of skew rhombic plates for varying angles of skew are not available in abundance and only some scattered result are found for some particular boundary conditions or for some particular angle of skew. In many of these works, again, values of moments which are important to a design engineer are not reported. Recently, Butalia have used the nine node heterosis element for the study of such plates for different angle of skew and support conditions.

B. Finite Element Methods (FEM)

The Finite Element Method (FEM) is an analyst choice of material model(s), finite elements meshes, constraint equations, analysis procedures, governing matrix equation and their solution methods, pre and post processing option implemented in a chosen commercial Finite Element Analysis (FEA) programme for the intended analysis of the structure or component. FEM in general, a common FEA programme in particular implemented on a personal computer offers a universal tool for engineering analysis.

FEM consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design, and existing product refinement. The user is able to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition.

There are generally two types of analysis they are: 2-D modeling, and 3-D modeling. While 2-D modeling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively. Within each of these modeling schemes, the programmer can insert numerous algorithms (functions) which may make the system behave linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture.

FEM uses a complex system of points called nodes which make a grid called a mesh. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions which will receive large amounts of stress usually have a higher node density than those which experience little or no stress. Points of interest may consist of: fracture point of previously tested material, fillets, corners, complex detail, and high stress areas. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties to the object, creating many Elements. An Element is a section of a body obtained from dividing the body up into a finite number of regions.

C. Overview of Plates and Shells

Thin-walled structures in the form of plates and shells are encountered in many branches of technology, such as civil, mechanical, aeronautical, marine, and chemical engineering. Such a widespread use of plate and shell structures arises from their intrinsic properties. When suitably designed, even very thin plates, and especially shells, can support large loads. Thus, they are utilized in structures such as aerospace vehicles in which light weight is essential.

Thin plates are initially flat structural members bounded by two parallel planes, called faces, and a cylindrical surface, called an edge or boundary. The generators of the cylindrical surface are perpendicular to the plane faces. The distance between the plane faces is called the thickness (h) of the plate. It will be assumed that the plate thickness is small compared with other characteristic dimensions of the faces (length, width, diameter, etc.). Geometrically, plates are bounded either by straight or curved boundaries. The static or dynamic loads carried by plates are predominantly perpendicular to the plate faces.

The load-carrying action of a plate is similar, to a certain extent, to that of beams or cables; thus, plates can be approximated by a grid work of an infinite number of beams or by a network of an infinite number of cables, depending on the flexural rigidity of the structures. This two-dimensional structural action of Plates results in lighter structures, and therefore offers numerous economic advantages. The plate, being originally flat, develops shear forces, bending and twisting moments to resist transverse loads. Because the loads are generally carried in both directions and because the twisting rigidity in isotropic plates is quite significant, a plate is considerably stiffer than a beam of comparable span and thickness. So, thin plates combine light weight and form efficiency with high load-carrying capacity, economy, and technological effectiveness.

Because of the distinct advantages discussed above, thin plates are extensively used in all fields of engineering. Plates are used in architectural structures, bridges, hydraulic structures, pavements, containers, airplanes, missiles, ships, instruments, machine parts, etc. We consider a plate, for which it is common to divide the thickness h into equal halves by a plane parallel to its faces.

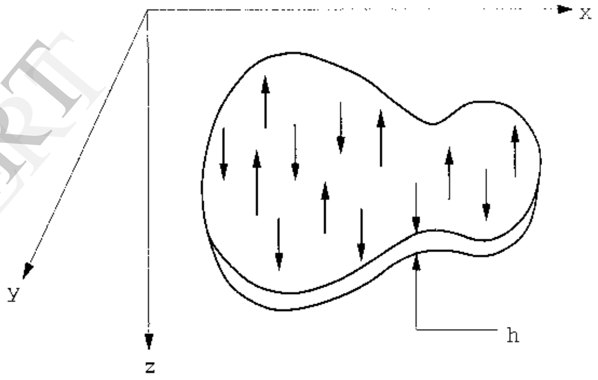


Figure 1: Plates are bounded either by straight or curved boundaries

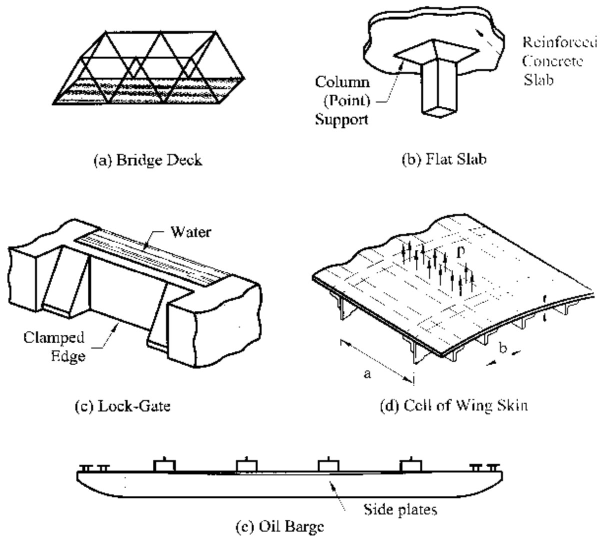


Figure 2: a) Bridge deck b) Flat slab c) Lock-gate d) Cell of wing skin e) Oil barage

This plane is called the middle plane (or simply, the mid plane) of the plate. Being subjected to transverse loads, an initially flat plate deforms and the mid plane passes into some curvilinear surface, which is referred to as the middle surface. We will consider only plates of constant thickness. For such plates, the shape of a plate is adequately defined by describing the geometry of its middle plane. Depending on the shape of this mid plane, we will distinguish between rectangular, circular, elliptic, etc., plates.

A plate resists transverse loads by means of bending, exclusively. The flexural properties of a plate depend greatly upon its thickness in comparison with other dimensions.

Plates may be classified into three groups according to the ratio a/h, where 'a' is a typical dimension of a plate in a plane and 'h' is a plate thickness. These groups are

- The first group is presented by thick plates having ratios a/h ≤ 8 . . . 10. The analysis of such bodies includes all the components of stresses, strains, and displacements as for solid bodies using the general equations of three-dimensional elasticity.
- The second group refers to plates with ratios a/h ≥ 80. . .100. These plates are referred to as membranes and they are devoid of flexural rigidity. Membranes carry the lateral loads by axial tensile forces N (and shear forces) acting in the plate middle surface. These forces are called membrane forces; they produce projection on a vertical axis and thus balance a lateral load applied to the plate-membrane.
- The most extensive group represents an intermediate type of plate, so called thin plate with 8. . . 10 ≤ a/h ≤ 80 . . . 100. Depending on the value of the ratio w=h, the ratio of the maximum deflection of the plate to its thickness, the part of flexural and membrane forces here may be different. Therefore, this group, in turn, may also be subdivided into two different classes.

a. Stiff plates. A plate can be classified as a stiff plate if w/h ≤ 0.2. Stiff plates are flexural rigid thin plates. They carry loads two dimensionally, mostly by internal bending and twisting moments and by transverse shear forces. The middle plane deformations and the membrane forces are negligible. In engineering practice, the term plate is understood to mean a stiff plate, unless otherwise specified. The concept of stiff plates introduces serious simplifications that are discussed later. A fundamental feature of stiff plates is that the equations of static equilibrium for a plate element may be set up for an original (unreformed) configuration of the plate.

b. Flexible plates. If the plate deflections are beyond a certain level, w/h≥0.3, then, the lateral deflections will be accompanied by stretching of the middle surface. Such plates are referred to as flexible plates. These plates represent a combination of stiff plates and membranes and carry external loads by the combined action of internal moments, shear forces, and membrane (axial) forces. Such plates, because of their favorable weight-to-load ratio, are widely used by the aerospace industry. When the magnitude of the maximum deflection is considerably greater than the plate thickness, the membrane action predominates. So, if w/h > 5, the flexural stress can be neglected compared with the membrane stress. Consequently, the load-carrying mechanism of such plates becomes of the membrane type, i.e., the stress is uniformly distributed over the plate thickness.

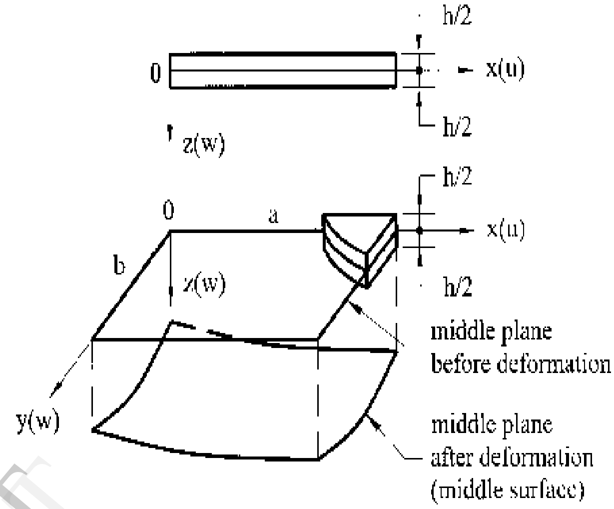


Figure 3: Mid plane

II. SKEW PLATES

Skew plates are widely used in modern structures. Swept wings of airplanes and parallelogram slabs in buildings and bridges are examples of the application of skew plates in engineering.

In the most general case the use of an oblique system of coordinate chosen in accordance with the given angle of skew should be recommended; in certain particular cases rectangular coordinates may also be used to advantage in dealing with skew plates, and the method of finite differences appears. The following numerical data for uniformly loaded skewed plates were obtained in that way.

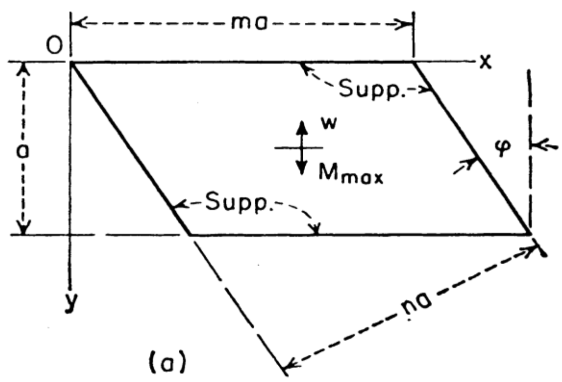


Figure 4: Skew plate with all the edges simply supported

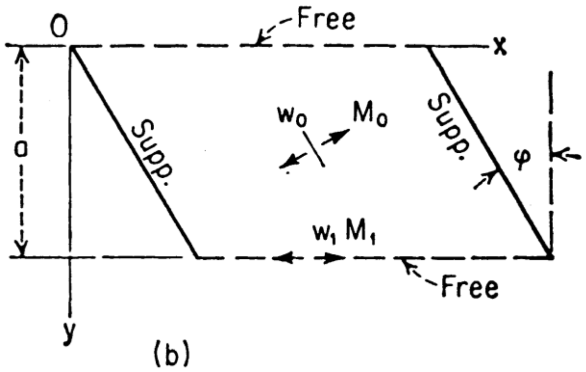


Figure 5: Skew plate with two edges is simply supported

III. Skew Plate Elements

A. PLANE82 Element Description

PLANE82 is a higher order version of the 2-D, four-node element. It provides more accurate results for mixed (quadrilateral-triangular) automatic meshes and can tolerate irregular shapes without as much loss of accuracy. The 8-node elements have compatible displacement shapes and are well suited to model curved boundaries.

The 8-node element is defined by eight nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element may be used as a plane element or as an axisymmetric element. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

PLANE82 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 6 "PLANE82 Geometry".

A triangular-shaped element may be formed by defining the same node number for nodes K, L and O. A similar, but 6-node, triangular element is PLANE2. Besides the nodes, the element input data includes a thickness (TK) (for the plane stress option only) and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems.

PLANE82 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output

The element stress directions are parallel to the element coordinate system. Surface stresses are available on any face. Surface stresses on face IJ, for example, are defined parallel and perpendicular to the IJ line and along the Z axis for a plane analysis or in the hoop direction for an axisymmetric analysis.

PLANE82 Assumptions and Restrictions

- The area of the element must be positive.
- The element must lie in a global X-Y plane as shown in Figure 6 "PLANE82 Geometry" and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the X quadrants.
- A face with a removed midside node implies that the displacement varies linearly, rather than parabolic ally, along that face

B. SHELL93 Element Description

SHELL93 is particularly well suited to model curved shells. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. The deformation shapes are quadratic in both in-plane directions. The element has plasticity, stress stiffening, large deflection, and large strain capabilities.

SHELL93 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 7. The element is defined by eight nodes, four thicknesses, and the orthotropic material properties. Midside nodes may not be removed from this element. A triangular-shaped element may be formed by defining the same node number for nodes K, L and O. Orthotropic material directions correspond to the element coordinate directions. The element x and y-axes are in the plane of the element. The x-axis may be rotated an angle θ (in degrees) toward the y-axis.

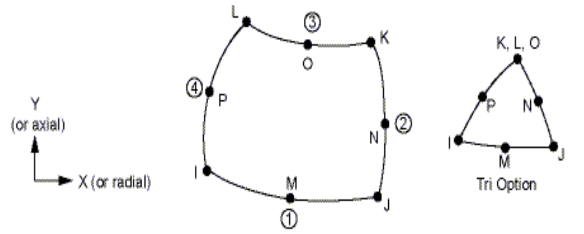


Figure 6: PLANE82 Geometry

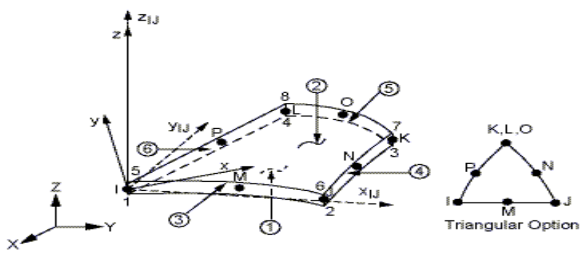


Figure 7: SHELL93 Geometry

ISSN: 2278-0181
Vol. 3 Issue 2, February - 2014

SHELL93 Output Data

The solution output associated with the element is in two forms:

- Nodal displacements included in the overall nodal solution
- Additional element output

SHELL93 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most often whenever the elements are not numbered properly.
- Zero thickness elements or elements tapering down to a zero thickness at any corner are not allowed.
- The applied transverse thermal gradient is assumed to vary linearly through the thickness.
- Shear deflections are included in this element.
- The out-of-plane (normal) stress for this element varies linearly through the thickness.
- The transverse shear stresses (S_{YZ} and S_{XZ}) are assumed to be constant through the thickness.
- The transverse shear strains are assumed to be small in a large strain analysis.
- This element may produce inaccurate stresses under thermal loads for doubly curved

IV. Finite Element Model Development

- MODEL 1 : Tapered skew cantilever plate/ cook's membrane plate
- MODEL 2 : Rhombic 30^0 skew plate
- MODEL 3: Rhombic 45^0 skew plate
- MODEL 4: Rhombic 60^0 skew plate

A. *MODEL 1: Tapered Skew Cantilever Plate/ Cook's Membrane Plate*

Coarse Model

The Geometry of the model is shown in Figure 8 and the material properties as shown in Table 1.The plate is clamped on one side while the opposite side is subjected to a distributed in-plane load at the free end. Finite element mesh model and boundary condition is obtained by using 8-node plane 82 quadrilateral element.

Table 1: Mechanical Material Properties

Material properties	Geometric detail	Applied load
Young's modulus=100N/mm^2 Poisson's ratio ν=0.33	Thickness, h=1.0mm	P=100N

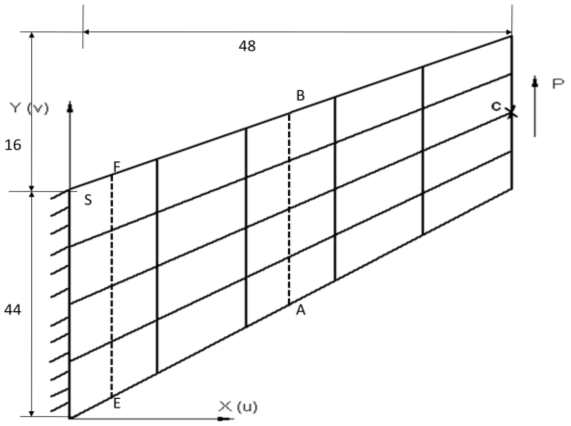


Figure 8: Geometry of Model 1

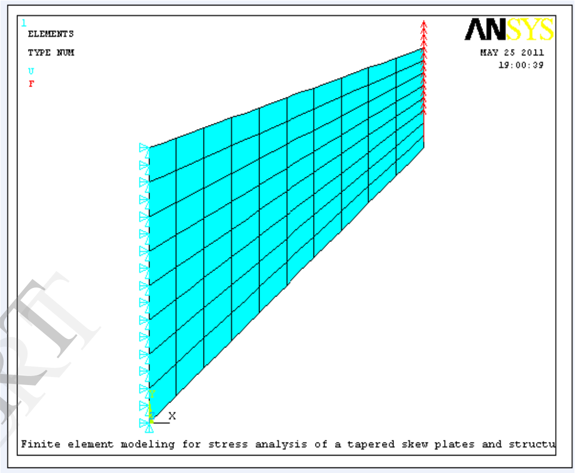


Figure 9: Finite Element Model and Boundary Condition of Model 1

B. *MODEL 2: Rhombic 30^0 skew plate*

Coarse Model

The Geometry of the Rhombic 30^0 skew plate model is shown in Figure 10, with simply-supported edges, subjected to uniform distributed pressure load P. The plate has thickness 'h' and side length 'b' and the mechanical properties of graphite/epoxy (or) steel are shown in Table 2. Static stress analysis is performed for two dimensional rhombic skew plate subjected to uniform distributed load P. Finite element mesh model and boundary condition is obtained by using 8-node shell 93 quadrilateral element.

Table 2: Mechanical Properties of Graphite/Epoxy (OR) Steel

Elastic modulus(E)	Poisson's ratio (ν)	Pressure load (P)
30×10^6MPa	0.3	1N

Table 3: Dimensions of Skew Plate

Length of the plate (a)	1.00mm
Width of the plate (b)	1.00mm
Thickness of the plate (t)	0.01mm
Angle of the plate (θ)	30^0

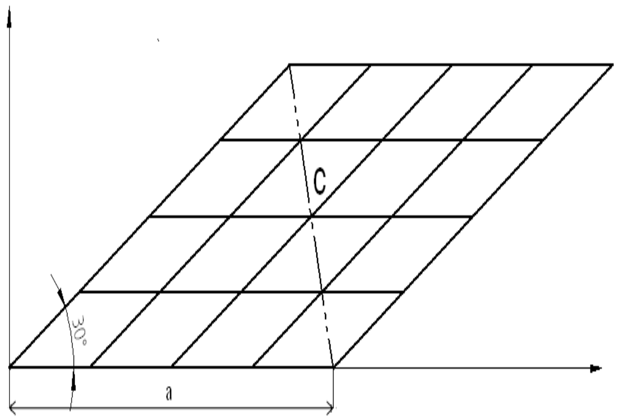


Figure 10: Geometry of Model 2

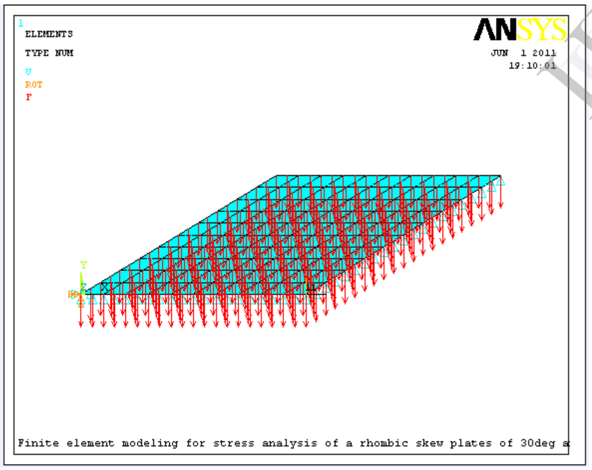


Figure 11: Finite Element Model and Boundary Condition of Model 2

C. *MODEL 3: Rhombic 45^0 skew plate*

The Geometry of the Rhombic 45^0 skew plate model is shown in Figure 12, with simply-supported edges, subjected to uniform distributed load P. The plate has thickness ‘h’ and side length ‘b’ and the mechanical properties of graphite/epoxy (or) steel are shown in Table 4. Static stress analysis is performed for two dimensional rhombic skew plate subjected to uniform distributed load P. Finite element mesh model and boundary condition is obtained by using 8-node shell 93 quadrilateral element.

Table 4: Mechanical Properties of Graphite/Epoxy (OR) Steel

Elastic modulus(E)	**Poisson’s ratio (ν)**	**Pressure load (P)**
30×10^6MPa	0.3	1N

Table 5: Dimensions of Skew Plate

Length of the plate (a)	1.00mm
Width of the plate (b)	1.00mm
Thickness of the plate (t)	0.01mm
Angle of the plate (θ)	45^0

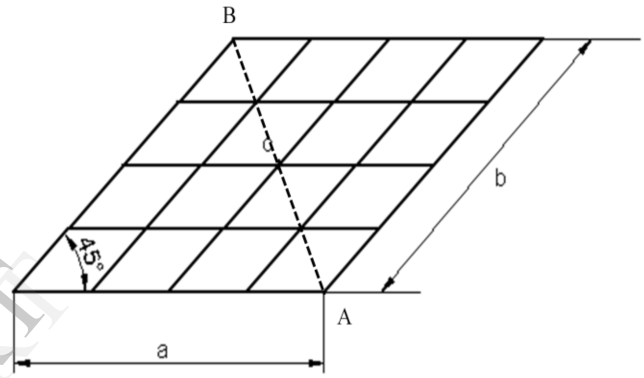


Figure 12: Geometry of Model 3

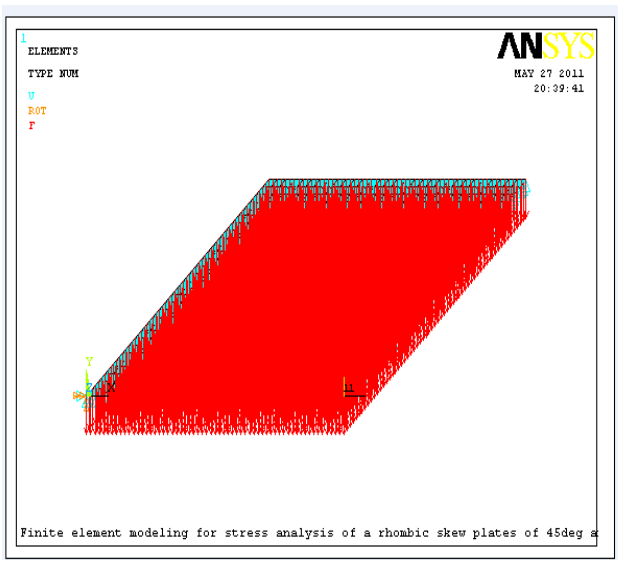


Figure 13: Finite Element Model and Boundary Condition of Model 3

D. MODEL 4: Rhombic 60^0 skew plate

The Geometry of the Rhombic 60^0 skew plate model is shown in Figure 14, with simply-supported edges, subjected to uniform distributed load P. The plate has thickness 'h' and side length 'b' and the mechanical properties of graphite/epoxy (or) steel are shown in Table 6. Static stress analysis is performed for two dimensional rhombic skew plate subjected to uniform distributed load P. Finite element mesh model and boundary condition is obtained by using 8-node shell 93 quadrilateral element.

Table 6: Mechanical Properties of Graphite/Epoxy (OR) Steel

Elastic modulus(E)	**Poisson's ratio (ν)**	**Pressure load (P)**
30×10^{6}Mpa	0.3	1N

Table 7: Dimensions of Skew Plate

Length of the plate (a)	1.00mm
Width of the plate (b)	1.00mm
Thickness of the plate (t)	0.01mm
Angle of the plate (θ)	60^0

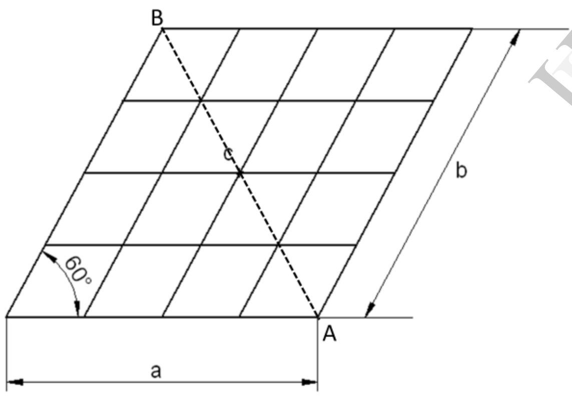


Figure 14: Geometry of Model 4

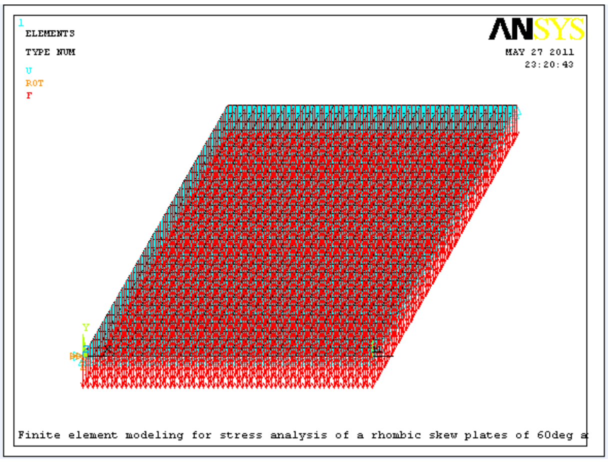


Figure 15: Finite Element Model and Boundary Condition of Model 4

V. FINITE ELEMENT MODEL VALIDATION

A. FEM Validation of Model 1: Tapered Skew Cantilever Plate

The Figure 16 shows the maximum deformation in Y-direction is 24.87 by applying the load 100N. The Figure 18 shows the maximum deformation in X-direction is 13.879 by applying the load 100N.

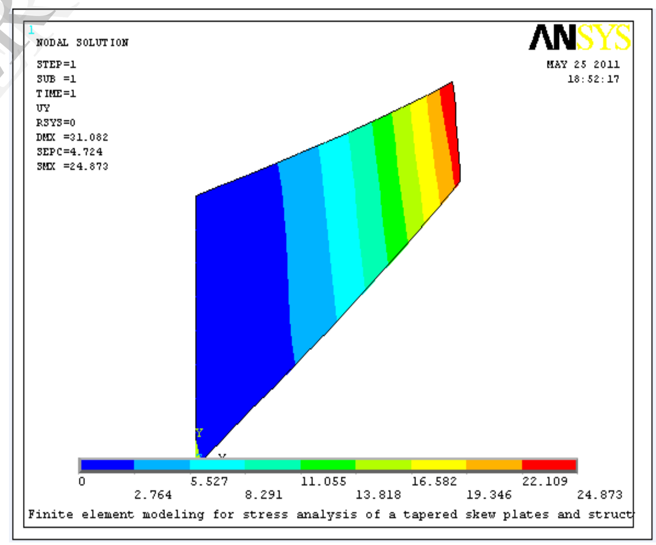


Figure 16: Deformation in Y-direction

The predicted displacement at point C in Y-direction as shown in Figure 8 is 23.718 while the targeted value is 23.91. For the same problem solving in ANSYS we achieve the value 23.932 as shown in Figure 17.

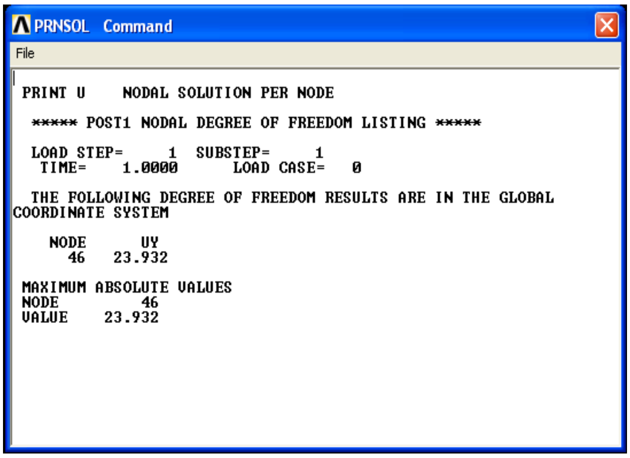


Figure 17: Displacement at point C in Y-direction result screen

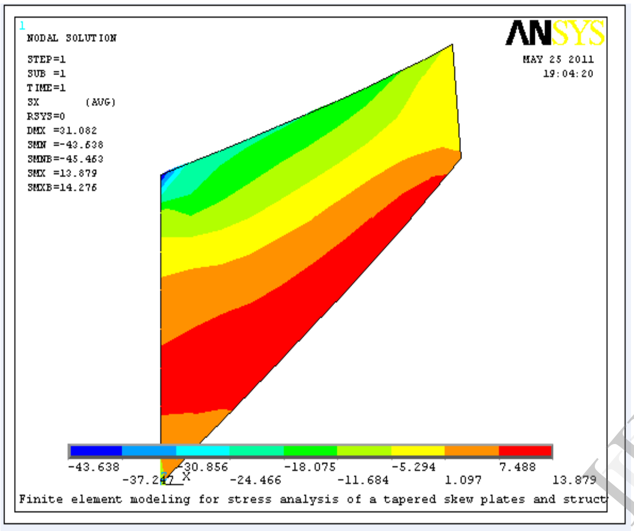


Figure 18: Stresses in X –direction

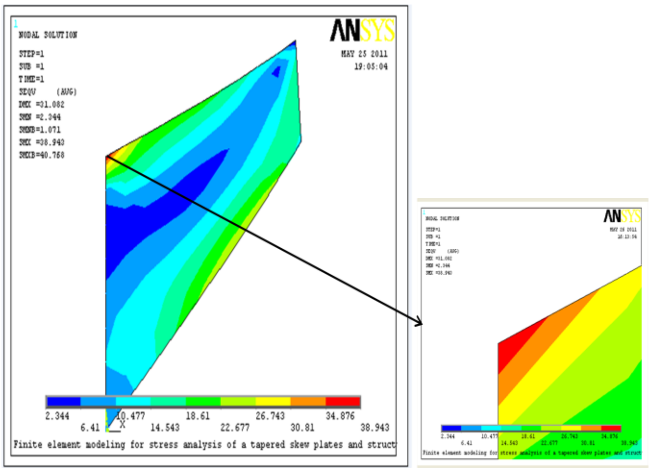


Figure 19: Maximum stresses in the corner

The Von Mises stress distribution for the coarse model of tapered skew cantilever plate is shown in Figure 19 in which the maximum stress is of 38.943. This is because there is a huge stress variation at the corner S shown in Figure 8. Sub modeling is done to capture the variation of stress at this corner.

B. FEM Validation of Model 2: Rhombic 30^0 Skew plate

The Figure 20 shows the X- Directional stress distribution in the model 2. The maximum stress variations are observed at the corners of obtuse angle as shown in zoomed view. Due to lager variation of stress at this corner, sub modeling is required to capture the variation of the stress in this region.

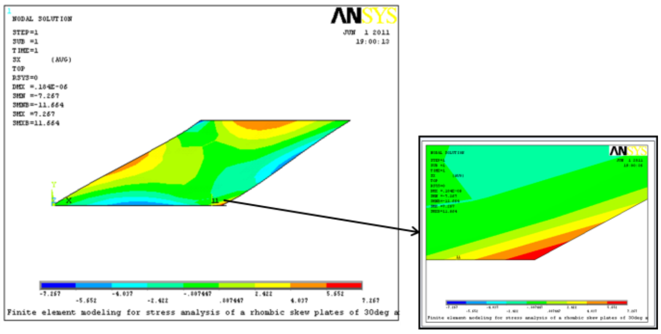
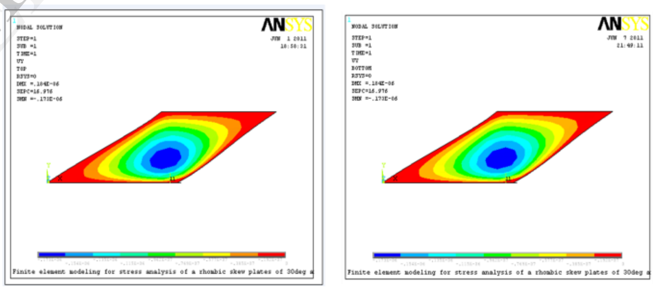


Figure 20: Stress in X-direction

From the Figure 21 the deflection in Y- Direction is shown for both top and bottom surfaces at point C. This concludes that one surface is at tension and another surface is in compression.



Top surface

Bottom surface

Figure 21: Maximum deformation in Y-Direction

C. FEM Validation of Model 3: Rhombic 45^0 Skew plate

For given specimen the centre deflection at point C shown in Figure 12 is evaluated by using different mesh sizes.

Von Mises Stress Distribution for the coarse model of rhombic 45^0 skew plats is as shown in Figure 22 in which the maximum stress is of 49.442.

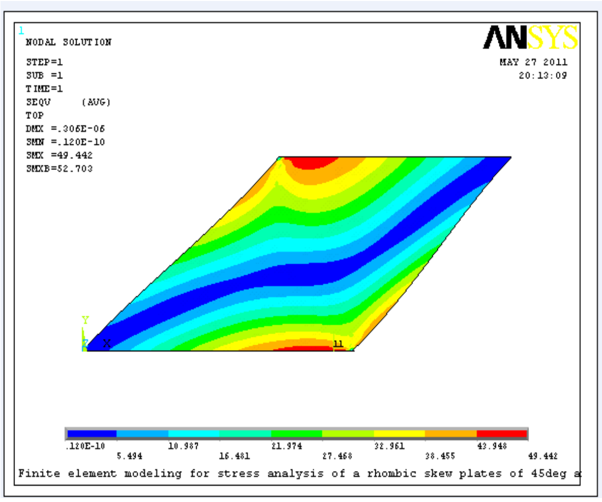


Figure 22: Von Mises Stress Distribution for Model 3

The Figure 23 shows the X- Directional stress distribution in the model 3. The maximum stress variations are not observed at the corners of obtuse angle as shown in zoomed view.

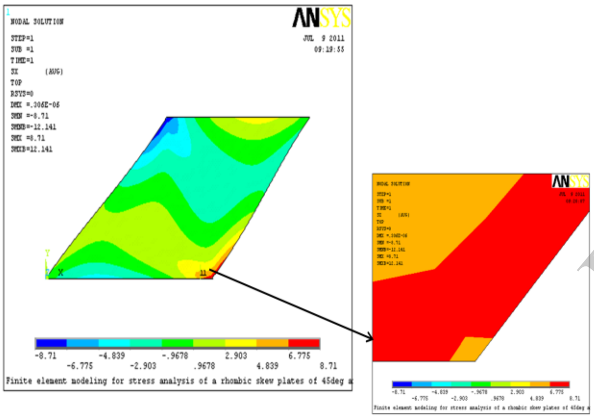
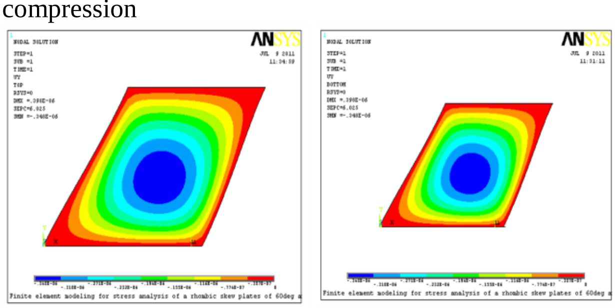


Figure 23: Stress in X-Direction

D. FEM Validation of Model 4: Rhombic 60^0 Skew plate

From the Figure 24 the deflection in Y-Direction is shown for both top and bottom surfaces at point C. This concludes that one surface is at tension and another surface is in compression



Top surface Bottom surface

Figure 24: Maximum deformation in Y-Direction

Von Mises Stress Distribution for the coarse model of rhombic 60^0 skew plats is as shown in Figure 25 in which the maximum stress is of 48.354.

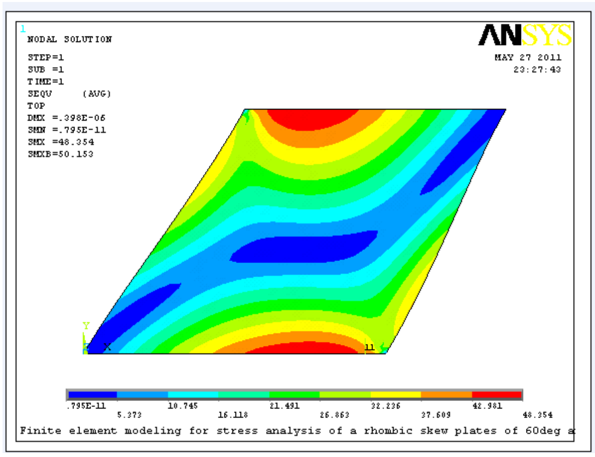


Figure 25: Von Mises Stress Distribution for model 4

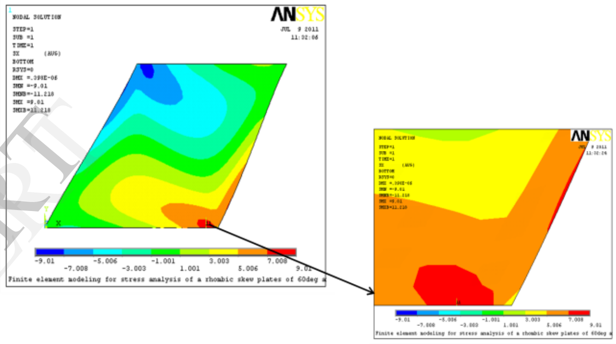


Figure 26: Stresses in X-Direction

The Figure 26 shows the stress distribution along the path AB for Top surfaces in X- Direction.

VI. CONCLUSIONS AND SCOPE

In the present study, the skew plate has been analyzed systematically by using Finite element method for uniform distributed load acting on the plate.

- The numerical results are displayed for different skew angles (30^0, 45^0 and 60^0) and the variation of von mises, X-Direction and Y-Direction stresses along the path AB is investigated.
- The validation of presented method is carried out successfully with the available result, thus established the accuracy of the present method.
- Sub modeling is used to capture the variation of stresses at corner for both tapered skew plate and rhombic skew plate of 30^0 angles.
- Comparing the results for rhombic skew plates of 30^0, 45^0 and 60^0, the large stress variation at corner is found to be for rhombic skew plate of 30^0.

- The present study provides an efficient analysis method for skew plate static stress analysis.

VII. Scope for Future work

- This work can be extended to various types of dynamic skew plate problems.
- This work can be extended by varying the aspect ratio.

References

[1] *H.V. Lakshminarayana and Kailash.K,* A shear deformable curved shell element of quadrilateral shape. Journal of Computers and Structures, Volume 33, Issue 4, 1989, pp. 987-1001,

[2] *H.V***.** *Lakshminarayana and S. Sridhara Murthy***,** A shear-flexible triangular finite element model for laminated composite plates. International Journal for Numerical Methods in Engineering, Volume 20, 1984, pp. 591-623.

[3] *T. S. Butalia, T. Kant and V. D. Dixit***,** Performance of heterosis element for bending of skew rhombic plates, Journal of Computers & Structures Volume 34, 1990, pp. 23-49.

[4] *M.P. Rossow***,** Efficient C^0 Finite Element Solution of Simply Supported Plates of Polygonal Shapes, Journal of Applied Mechanics, June 1977, pp. 347-349.

[5] *Debabrata Das, Prasanta Sahoo and Kashinath Saha***,** A variational analysis for large deflection of skew plates under uniformly distributed load through domain mapping technique, International Journal of Engineering, Science and Technology, Volume 1, No. 1, 2009, pp. 16-32.

[6] *G.R. Liu, T. Nguyen-Thoi, K.Y. Lam***,** A novel alpha finite element method (άFEM) for exact solution to mechanics problems using triangular and tetrahedral elements, Computer Methods in Applied Mechanics and Engineering, Volume 197, 2008, pp. 3883-3897.

[7] *M. Duan and M. Mahendran***,** Large deflection analyses of skew plates using hybrid/mixed finite element method, Journal of Computers & Structures, Volume 81, Issue 13, May 2003, pp. 1415-1424.

[8] *Hota V.S. GangaRao and V.K. Chaudhary***,** Analysis of skew and triangular plate in bending, Journal of Computers & Structures, Volume 28, No. 2, 1988, pp. 223-235.

[9] *D. Sengupta***,** Performance study of a simple finite element in the analysis of skew rhombic plates, Journal of Computers & Structures, Volume 54, No. 6, 1995, pp. 1173-1182.

[10] *H.V.Lakshminarayana***,** Finite Element Analysis and Procedures in Engineering (2010).

[11] *Timoshenko.S.P and Woinowosky-Krieger.S***,** Theory of plates and shells, 2nd Edition, McGraw-Hill, New York (1956).