

Optimizing The Best Angle Of The Grains For Composite Material Using Fem

V.M.SIDDESH¹, S.SURESH²

¹PG Student, CAD/CAM, Siddharth Institute of Engineering & Technology (SIETK), Puttur - 517583

²Department of Mechanical Engineering, SIETK College, Puttur, Chittoor Dist, Andhra Pradesh, INDIA

ABSTRACT

The project was titled as "OPTIMIZING THE BEST ANGLE OF THE GRAINS FOR COMPOSITE MATERIAL USING FEM". Composite material consists of two constituents mainly fibers and matrix. Generally all the materials are made of the grains with different grain alignments. In these project 10 different materials are used each of different grain angles and different thickness. In this each material was divided into 8 layers. Analyzing the component can get the "Best angle" and the "Best thickness" alignment of the grains. The analysis is made by using ANSYS 8.0. The element used in project was Shell 99. Main advantages of composite material are high strength.

The composite materials are generally used in Aero space vessels, using in the manufacturing of Golf sticks etc.,

Keywords – Aero space vessels, Golf sticks

1. INTRODUCTION TO COMPOSITE:

1.1 Definition and properties of composite material

The composite material consists of two or more materials combined on a macroscopic scale to form a useful material. The key is the macroscopic examination of a material. Different materials can be combined on a macroscopic scale as in alloying, but the result material is macroscopically homogeneous.

The advantage of the composites is that they usually exhibit the best qualities of their constituents and often some quantities that neither constituent possesses. The properties that can be improved by forming a composite include.

1. Strength
2. Stiffness
3. Corrosion resistance
4. Wear resistance
5. Attractiveness
6. Weight
7. Fatigue life
8. Temperature- dependent behavior
9. Thermal insulation
10. Thermal conductivity
11. Acoustical insulation

1.2 Need for composites

Aeronautical engineering is a very sophisticated field in which designers have to work with very low factor of safety. The requirements and specifications spelt out in aeronautical practice are extremely stringent, maximum reliability and safety. The design process obviously calls for highest human and materials standards, combined with external vigilance.

In order to meet the above stringent requirements, an air crafts designers has to continuously cope up with increasing demand on improved performance and better utilization of air crafts, due to tremendous advances in the fronts of increased speeds, higher altitudes, increased ranges, increased pay loads increasing maneuverability and STOL\VTOL air craft.

Classification of composite materials:

Classification the composite in the following three commonly accepted types

- 1) Fibrous composites
- 2) Laminated composites
- 3) Particulate composites

1. Fibrous composites:

- This composite consists of fibers in a matrix where matrix is a binding material.
- The fiber may be short or discontinues and randomly arranged; continuous filaments arranged parallel to each other.

2. Laminated composites:

- Two or more layers of material bonded together from a laminated composite. Laminates are made by stacking layers of mats or woven fabric, which have many possible weave configurations. The arrangement of layers of a laminate is known as the stacking sequence.
 - Laminated can be classified as
 - Unidirectional laminates
 - Bi-directional laminates
 - Multi directional laminates

- Plywood is a laminate consisting of thin sheets of wood arranged with the grain in alternate sheets at right angles.
- Common examples of laminates are in automobile windshields (laminated glass) and bimetal thermostats.

In both cases homogeneous, isotropic layers of materials are bonded, together to form non-homogeneous composite laminates.

3. Particulate composites

- They are composed of particles in a matrix. These composites encompass a wide range of materials.
- The reinforcing phase is often spherical or at least has dimensions of similar order in all directions.
- Solid rocket propellants represents a very special case of particulate composite in which inorganic propellant particles, about 75% by volume are bounded in an organic matrix such as polyurethane.
- Common examples of particulate composites are concrete, filled polymers, solid rocket propellants, and metal and ceramic particles in metal matrices.

II. The Finite Element Method:

Virtually every phenomenon in nature whether biological, geological, or mechanical can be described with the aid of laws of physics, in terms of algebraic, differential or integral equations relating various quantities of interest.

While the derivation of the governing equations for these problems is not unduly difficult their solution by exact methods of analysis is a formidable task. In such cases approximate methods of analysis like finite element method (FEM) provides alternative means.

The FEM is used to find approximate solutions to those problems, whose complex governing equations and boundary conditions prohibit their solutions, by the classical methods. It is a powerful numerical procedure for solving the mathematical problems of engineering and physics.

The model is composed of a set of piece wise continuous functions defined over a finite number of sub-domains, in other words, FEM, indicates that every physical system can be thought of as consisting of several components or members or sub-members, each components contributing to the behavior of the system as a whole.

FEM equations can be derived using weighted residual methods like gallerkins method, variational method or finite difference approximation.

1. Field conditions
2. Boundary conditions
3. Approximate solutions

3. Process of Finite Element Method:

In the field of engineering we often that it is necessary to obtain approximate numerical solutions to problems rather than exact closed form solutions. The finite element method is an approximate numerical method for solving the differential equations in physics and engineering. This method was originally introduced by Argyris, who published a series of papers on this and by tuner, coulgh, martin and top in paper published in 1956.

FEA relies strongly on matrix formulation of structural analysis. It has developed simultaneous with the growing emphasis on numerical methods for engineering analysis. This method started as a generalization of the structural idea to some problems of elastic continuum problems, started in terms of differential equations or as an extrimum problem.

3.1 Process for analyzing a structure:

3.3.1) Initial thinking

Quite often the time fir taken for an analysis can be minimized if the analysis makes a detailed study of the problem to be solved before touching a computer. In this initial thinking stage the analyst needs to consider the structural problem and understand as much as possible. Information such as geometrical shape of the component, its expected structural behavior.

When subjected to loads, typical applied loads & materials properties are particularly important. To gather this information liaison is necessary with variety of people such as design engineers, process engineering, material specialists & technicians. Further information can be gained from a literature search of the precious work.

3.3.2 Mesh generation

This second stage involves the production of a computer model of the geometry of the component.

The geometry is discretized to form a mesh of element and nodes. This means that the structure is broken down into small sub-regions on which the numerical analogue of the governing equations can be developed.

3.4 Specification of numerical problem

In the initial thinking stage, relevant boundary conditions have to be thought of in the context of the physics of the problem. Since the nodes & elements have been created in the mesh generation stage these boundary conditions must be reinterpreted in terms of the mesh. This will involve the specification of, for example, the forces to be applied at certain nodes, the pressure at certain elements faces and the known displacements at given nodes.

Similarly initial conditions may also have to be specified in more complex calculations. Finally the material properties of all the elements will need to be defined.

3.5 Solution of the equations

The solution involves solving for the displacement and then obtaining the stress and strains. Various efficient solution algorithms are available which can give accurate and fast solutions.

Definition of problem:

The failure of composites has been investigated extensively from the micromechanical and macromechanical points of view. On the micromechanical scale, failure mechanisms and processes vary widely with type of loading and are intimately related to the properties of the constituent phases, i.e., MATRIX, reinforcement, and interface interphase. Failure predictions based on micromechanics, even when they are accurate with regard to failure initiation at critical points, are only approximate with regard to global failure of a lamina and failure progression to ultimate failure of a multi-directional laminate. For these reasons a macro mechanical approach to failure analysis is preferred.

The different ways in which curing residual stresses are introduced in the predictions, especially in the case of first-ply-failure.

5. FEA Software - ANSYS

The abbreviation of ANSYS is Analysis System. Dr. John Swanson founded ANSYS in 1970 with a version to commercialize the concept of the computer simulated engineering, establish himself as one of the pioneer of finite element analysis (FEA).

ANSYS has evolved into multipurpose design analysis software ANSYS has evolved into multipurpose design analysis software program is externally powerful and easy to use. Each release hosts new and enhanced capabilities that make the program more flexible.

Overview of the program

The ANSYS program is a flexible robust design analysis and optimization package. The software operates on major computers and operating systems, from PCs to workstations to supercomputers.

ANSYS features file computability throughout the family of product and across all platforms. ANSYS design data access enables user to import computer-aided design models into ANSYS, eliminating repeat work. Although the ANSYS program has extensive and complex capabilities, its organization.

The ANSYS program allows engineering to construct computer models or transfer CAD models of structures, products, components, or systems; apply operating loads or other design performance conditions and study physical responses, such as stress levels, temperature distributions, or other design stress levels, temperatures distributions, or the impact of the impact of electromagnetic fields.

ANSYS is finite elements analysis software that allows engineers to perform the following tasks with ease.

- ✓ Build computer models or transfer CAD models, components or systems.
- ✓ Apply operating loads or other design performance conditions.
- ✓ Study the physical response such as stress levels, temperatures distribution etc.,
- ✓ Optimizing a design early in development to reduce production costs.

Organization of ANSYS

ANSYS is organized into two basic levels:

1. Begin level.
2. Processor level.

Begin level act as gateway in and out of the ANSYS program. It is also used for certain global program control such as, changing job names & clearing database. In order to analyze a model in the ANSYS environment the following steps are followed.

- 1) Preprocessing.
- 2) Processing
- 3) Post processing

1. Pre processing stage

The pre processing stage consists of specifying, the element type to be used, material properties & real constants. After specifying these, above said parameters, the user begins to model the components.

2. Processor Stage

This stage usually follows the pre processing stage. After the model is completed and the boundary conditions are applied the system is solved.

3. Post processor stage

This stage follows the processor stage. In this stage the results obtained in the processor stage are displayed on the graphic window & the results can be analyzed.

ANSYS Program features

The program features are listed below.

- ✓ Direct link between structural output & thermal input for thermal stress analysis.
- ✓ Graphics based user interface with mouse-driven menu system and comprehensive online help.
- ✓ Solid modeling for geometry definition, load application & automating mesh generation.
- ✓ Extensive graphic capabilities including color, hidden line removal, section, contour, variable versus path displays & (x, y) graphs.
- ✓ Parametric language for design studies, parametric solid modeling and design optimization.
- ✓ Design optimization including shape optimization.
- ✓ Mapping of analyses results into a path either in 2-d or 3-d space allowing scalar and vector calculus operations on the path.
- ✓ Operations include calculation of derivatives, integrals, and normal, tangent and position vectors.
- ✓ Energy error norm calculation.
- ✓ Sub-modeling to allow re-analyzing apportion of a modal using a much finer mesh.
- ✓ Fracture and fatigue calculations.
- ✓ User-created elements, files and routines.
- ✓ Interfaces to many other programs such as CAD systems.
- ✓ Geometric nonlinear effects model.
- ✓ Cables and compression- only spars.
- ✓ Interfaces (gaps).
- ✓ Control switching elements.
- ✓ Cracking and or crushing solids.
- ✓ Large deflections.
- ✓ Stress stiffening.
- ✓ Spin softening.
- ✓ Material non linear effects model.
- ✓ Plasticity.
- ✓ Hyper plasticity.
- ✓ Creep, swelling.
- ✓ Nonlinear magnetic.
- ✓ Temperature dependant properties.
- ✓ Phase change.

ANSYS Analysis Program:

ALGORITHM

Step1: Specify title“Best angle in composite material”

Step2: Definition of analysis type“Structural analysis”

Step3: Definition of element type“SHELL 99”

Step4: Define real constants the layers placed is 80

Step5: Define material properties Model > Material prop > Isentropic Enter material properties

Step6: Model creating Draw key points >join with lines >make area

Step7: Meshing Mesh tool >Area >size of the mesh

Step8: Applying loads
Solution>apply\loads>structural>Displacement Solution >apply loads>structural >force

Step9: Analyzing the model Post Processor >nodal solution

Methodology

Stage 1: In the pre processor stage of ANSYS consists of

- Type of analysis.
- Element type.
- Real constants.
- Modeling of the component.
- Meshing the component.

1. Type of analysis: The ANSYS consists of so many type of analysis in that we are choosing the structural analysis.

2. Element type: The element choosing in this project was SHELL99 this very best layer it having facility to place 250 layers.

3. Real constants: Real constants for this project were no layers placed were 80 and each was having constant thickness and also having same material properties.

Material no	Angle	Theta
1	0	0.1
1	45	0.25
1	60	0.5
1	90	1
1	0	1
1	45	0.5
1	60	0.25
1	90	0.1

TABLE: 1

These values are repeated for the 10 material as given combination.

5. Modeling the component:

Draw the key points on inactive cs and join them with the lines. The lines made into area.

6. Meshing the component:

The component was created by using the modeling that component was meshed with the meshing tool. In that mesh tool by choose the area set. By setting the area it will ask the size the element the size was given as 1 unit then becomes as finite elements.

By meshing the component the total preprocessor was completed.

Stage 2

In the processor stage of ANSYS consists of

1. Apply loads
 - Displacement
 - Force

2. Solution

1. Apply loads:

The applications loads was done on the component by using the solution in that

- **Displacement:**

In this displacement loads were applied on the nodes these nodes selected by the box zoom. And all the degrees of freedom are arrested by displacement.

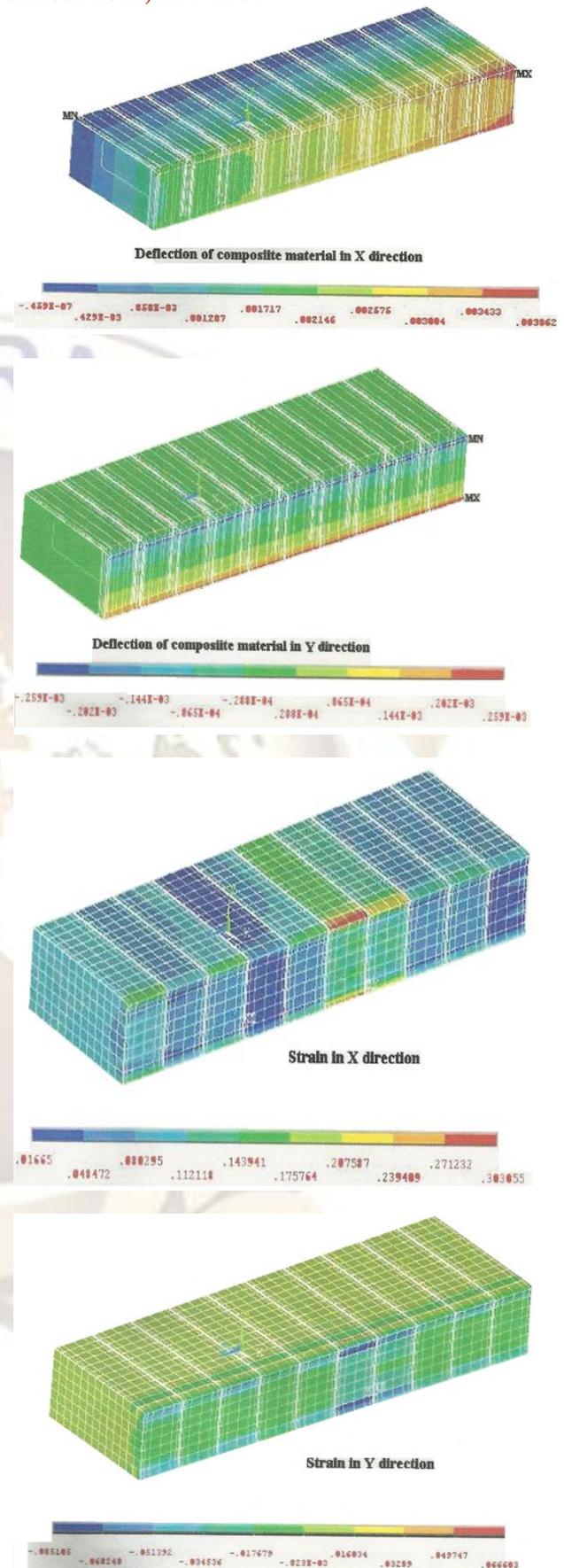
- **Force:**

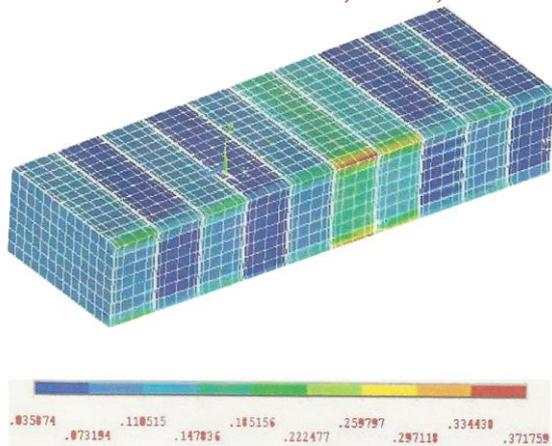
The other side of apply the force that was going towards the direction of the “X” in that direction apply the force of 1 unit was applied.

2. Solution:

After applying the loads the current solution was solved. By using the command **solve current LS.**

Stage 3 In the post processor stage of ANSYS choose the general post processor in that see the plot result and then nodal solution for seeing the deflection and other properties of the component after applying the load.





- ✓ This element may produce inaccurate results under thermal loads for non-flat domains.
- ✓ The applied transverse thermal gradient is assumed to be linear through the element and over the element surface.
- ✓ The stress varies linearly through the thickness of each layer.
- ✓ Interlaminar transverse shear stresses are based on the assumption that no shear is carried at the top and bottom surfaces of an element.

Further, these interlaminar shear stresses are only computed at the centroid and are not valid along the element boundaries. If accurate edge interlaminar shear stresses are required, shell-to-solid submodeling should be used.

4.7 Element used

A. Element Description

The SHELL 99 was “linear layered structural shell.” SHELL99 is used for layered applications of a structural shell model. While SHELL99 does not have some of the nonlinear capabilities of SHELL91, it usually has a smaller element formulation time. SHELL99 allows up to 250 layers. If more than 250 layers are required, a user-input constitutive matrix is available.

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. See SHELL99 in the ANSYS, Inc. Theory Reference for more details about this element.

Properties of SHELL 99

Nodes I, J, K, L, M, N, O, P

Degrees of Freedom UX, UY, UZ, ROTX, ROTY, ROTZ

B. SHELL99 Assumptions and Restrictions

- ✓ Zero area elements are not allowed. This occurs most often whenever the elements are not numbered properly.
- ✓ Zero thickness layers are allowed only if a zero thickness is defined at all corners. Tapering down to zero is not allowed.
- ✓ If KEYOPT (11) = 0, all nodes are assumed to be at the mid thickness of the element.
- ✓ The offset effect of the layers from the node is automatically included.
- ✓ No slippage is assumed between the element layers. Shear deflections are included in the element, however, normal to the center plane before deformation are assumed to remain straight after deformation.

The element matrices are reformed every iteration unless option 1 of the KUSE command is active. Only the lumped mass matrix is available. The mass matrix is assumed to act at the nodal plane.

The large deflection option for SHELL99 is not as convergent as it is for SHELL91 (the nonlinear layered shell element). SHELL91 may be the preferred element type when constructing models that include large deflection.

RESULTS AND CONCLUSION

Results:

In this project structural analysis was conducted on the composite materials by using ANSYS 8.0. Boundary conditions are all the degrees of freedom are arrested in one direction and other side the force as given in U_x direction. The force given is 1 N/m^2

The composite materials used in the project are

S.no	Material name	Young's modulus (N/mm ²)
1	Glass fiber reinforced	150
2	Silicon nitrate	300
3	Steel	210
4	Hot pressed silicon carbide	430
5	High carbon composites	560
6	Aluminum nitrate	320
7	Carbon composites	180
8	Aluminum oxide	380
9	Translucent polycrystalline	290
10	High carbon composites	510

CONCLUSION:

The experimental results are tabulated for each material; the component was prepared by using two

different composite materials. By comparing all the results, we choose fourth material was best material. Because it having less deformation less strain.

The fourth material was Hot pressed silicon carbide. All the grain angles are best for this material. This material gives best strength when compare to other materials.

BIBLIOGRAPHY

- Barton M.V., "Fundamentals of Aircraft Structures", prentice Hall, New York, 1948.
- Hoch berg, "Composite Material in Engineering Design", Edward brother Inc., Michigan, 1973.
- Lee.R.Cacolate, "The analysis of laminated structures", Van Nostard Reinhold Co., 1969.
- Robert M.Jhones., "Mechanics of Composite Material", McGraw-Hill, kogakusha Ltd., Tokyo.1975

AUTHORS



V.M.SIDDEESH was born in India in 1982. He received B.E. degree from ANNA UNIVERSITY and Also M.B.A. degree from PONDICHERRY UNIVERSITY. He Worked as a Project Engineer in HI-TECH HYDRAULICS PVT.LTD., Hyderabad, and also as a Production Engineer in FOODS & INNS LTD, Chittoor. Now at present doing his M.Tech (CAD/CAM).



S.SURESH was born in india 1982.He received B.Tech degree from VITS, kavali in 2004, M.Tech degree from S.V.University in 2008.since 2008 has been working in till date Mechanical Department at Siddharth Institute Of Engineering and Technology, Puttur as Associate Professor, Now Head of Aeronautical Engineering.