

Design And Analysis Of Carbon Fibre/Epoxy And Glass Fibre/Epoxy Pultruded Rods

Inzamamul Haq¹, A.Vidya sagar²

¹P.G. Scholar, ²Guide, Head of the Department

^{1,2} Branch: Mechanical (Machine Design)

^{1,2} Geethanjali College of Engineering and Technology

Email: ¹inzamamarif@gmail.com

Abstract:

Pultrusion is a continuous process for manufacture of composite materials with constant cross-section. Reinforced fibers are pulled through a resin, possibly followed by a separate performing system, and into a heated die, where the resin undergoes polymerization. Many resin types may be used in Pultrusion including polyester, polyurethane, vinyl ester and epoxy. This project deals with production of composite rods of both glass fiber with epoxy and carbon fiber with epoxy by using Pultrusion technique. The composition of composite rod contains of 85% fiber loading and 15% of matrix. This project also extends for modeling and analysis of composite rods. 3d model of composite rods is generated by using Unigraphics software. Analysis can be done by using ANSYS software. Composite rods are analyzed for different materials (i.e. Glass/Epoxy and Carbon/Epoxy materials.) Results from Analysis are compared for better material.

Keywords:

INTRODUCTION OF COMPOSITES:

A composite is usually made up of at least two materials out of which one is the binding material, also called matrix and the other is the reinforcement material (fiber, Kevlar and whiskers). The advantage of composite materials over conventional materials are largely from their higher specific strength, stiffness and fatigue characteristics, which enables structural

design to be moreover satile. By definition, composite materials consist of two or more constituents with physically separable phases. However, only when the composite phase materials have notably different physical properties and it is recognized as being a composite material. Composites are materials that comprise strong load carrying material (known as reinforcement) imbedded in weaker material (known as matrix).

Reinforcement provides strength and rigidity, helping to support structural load. The matrix or binder (organic or inorganic) maintains the position and orientation of the reinforcement. Significantly, constituents of the composites retain their individual, physical and chemical Properties; yet together they produce a combination of qualities which individual constituents would be in capable of producing alone. The reinforcement may be Platelets, Particles or fibers and are usually added to improve mechanical Properties such as stiffness, strength and toughness of the matrix material. Long fibers that are oriented in the direction of loading offer the most efficient load transfer. This is because the stress transfer zone extends only over a small part of the fiber-matrix interface and perturbation effects at fiber ends may be neglected. In other words' the ineffective fiber length is small. Popular fibers available as continuous filaments for use in high performance composites are glass, carbon and aramid fibers

APPLICATION OF FIBER COMPOSITES:

Today with all the advancement in life, growing population and its increasing needs we have to consider how to meet those needs by producing the best and making it available for all. Fiber reinforced composites are helping to fulfill the needs of this growing population. They have been and are being studied in order to maximize their utilities in different fields. This part of the chapter will look into the various fields that fiber reinforced composites are being used and what are their environmental impacts.

Application of fiber composites are represented by the following groups which are 70% of the total market value: automotive (23%), building and public works (21%), aeronautics (17%) and sports (11%).

INTRODUCTION TO UNIGRAPHICS:

NX is one of the world's most advanced and tightly integrated CAD/CAM/CAE product development solutions. Spanning the entire range of product development, NX delivers immense value to enterprises of all sizes. It simplifies complex product designs, thus speeding up the process of introducing products to the market.

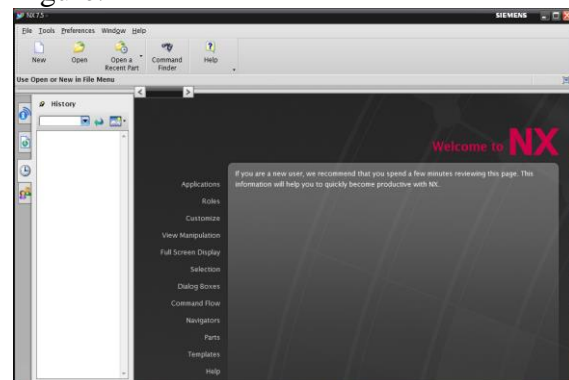
The NX software integrates knowledge-based principles, industrial design, geometric modeling, advanced analysis, graphic simulation, and concurrent engineering. The software has powerful hybrid modeling capabilities by integrating constraint-based feature modeling and explicit geometric modeling. In addition to modeling standard geometry parts, it allows the user to design complex free-form shapes such as airfoils and manifolds. It also merges solid and surface modeling techniques into one powerful tool set. Our previous efforts to prepare the NX self-guiding tutorial were funded by the National Science Foundation's Advanced Technological Education Program and by the Partners of the Advancement of

Collaborative Engineering Education (PACE) program.

NX is a premier 3D computer aided design suite. It allows you to model solid components and assemblies, to perform engineering analyses such as mechanism simulation and stress analysis, to create tool paths for computer-based manufacturing processes and to perform numerous other engineering design activities in a single software environment. Software suites like NX are referred to as product lifecycle management (PLM).

GETTING STARTED

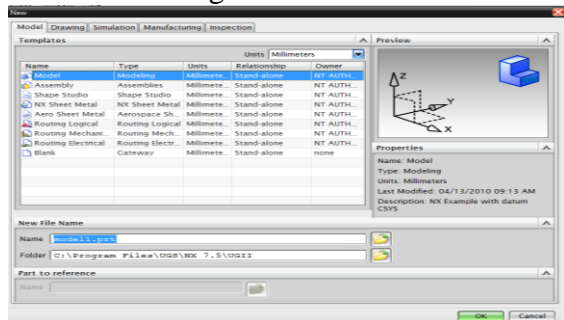
We begin with starting a NX7.5 session. This chapter will provide the basics required to use any CAD/CAM package. You will learn the preliminary steps to start, to understand and to use the package for modeling, drafting, etc. To start NX, use the NX 7.5 shortcut under Start (or Windows button)->(All) Programs->UGS NX 7.5. It may take a minute or so for NX to start the first time. Once opened, you will be presented with the window shown in below Figure.



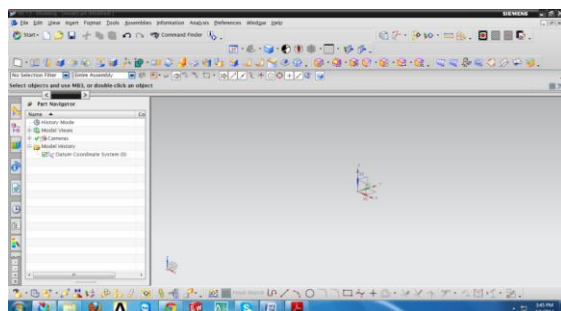
STARTING A NEW MODEL

To start modeling, you must first create a part file. NX part files use the extension .prt for both components and assemblies of components. In this tutorial, a single part file will be used with one component. To create a new part, click the New button to open the New dialog. For now, stay in the Model tab. Select Model from the Templates list (the default) and set the Folder to a location on your S: drive or

desktop. The default location may be stored locally on the lab computer you are using and might not be in your roaming profile (it might not be accessible on a different computer). Once a folder has been selected, set the name of the part file. Click OK to start modeling.



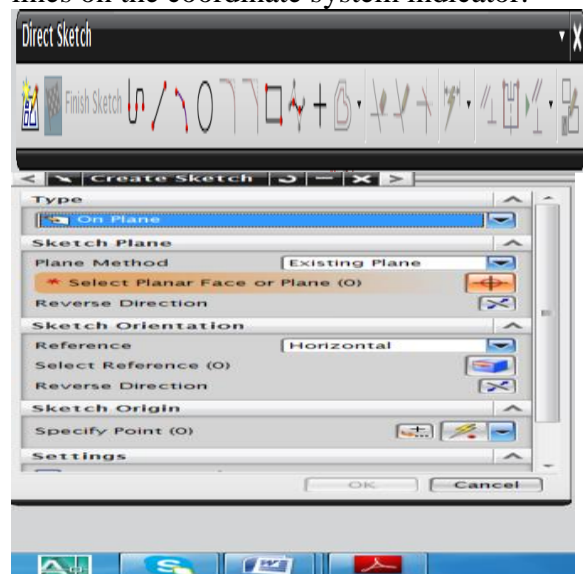
Once the new file has been created, the NX modelling interface will open (Figure 3). Like most modern PLM tools, the interface for NX contains numerous icons, lists, text prompts and other features that can be incredibly overwhelming. For now, we will focus on the sketching tools, part navigator, viewer and menu.



DRAWING IN NX

NX, like most modern PLM tools, is feature-based. That means you build up a component from a set of features that are added in sequence. This sequence, and the details of each feature, can be edited later if the design needs to be changed. Features can be removed, inserted or modified to create almost any solid part imaginable. This allows a precise and highly editable method for describing solid components and differs from the 3D freeform modeling approach (freeform modeling is used in computer graphics applications such as making models

for video games). The features you add to a part will appear in the Part Navigator under Model History. For a new part, the only thing appearing in the model history should be a Datum Coordinate System. This is the 3D Cartesian coordinate system used in the part and includes an origin location and coordinate axes. These are shown with the coordinate system indicator at the center of the Viewer. The general workflow for creating a part in NX is to create a sketch, extrude or revolve the sketch and then add features to the resulting solid part. Sketches are 2D (mostly) drawings that are used to define a cross section of the part. This cross section can be extruded (pushed into or pulled out of the plane of the sketch) and rotated (wrapped around an axis) to form a 3D solid part. To create a Sketch. Click the Sketch button in the sketching toolbar. This will open a new dialog for defining the sketch. The first step in defining the Sketch is to set the sketching plane. This is the 2D plane that the sketch will be drawn onto. This plane can be derived from the existing Datum Coordinate System or can be based on geometry from an existing part (you can sketch on the surface of a part). For now, start the sketch on the XY plane. To select the XY plane, move the cursor to the skewed lines connecting the x and y axis lines on the coordinate system indicator.



NX, also known as NX Unigraphics or usually just U-G, is an advanced CAD/CAM/CAE software package developed by Siemens PLM Software.

It is used, among other tasks, for:

- Design (parametric and direct solid/surface modeling)
- Engineering analysis (static, dynamic, electro-magnetic, thermal, using the Finite Element Method, and fluid using the finite volume method).
- Manufacturing finished design by using included machining modules First release of the new "Next Generation" version of Unigraphics and Ideas, called NX. This will eventually bring the functionality and capabilities of both Unigraphics and I-DEAS together into a single consolidated product.

Increasing complexity of products, development processes and design teams is challenging companies to find new tools and methods to deliver greater innovation and higher quality at lower cost. Leading-edge technology from Siemens PLM software delivers greater power for today's design challenge. From innovative Synchronous Technology that unites parametric and history-free modeling, to NX Active Mockup for multi-CAD assembly design, NX delivers breakthrough technology that sets new standards for speed, performance, and ease of use.

NX automates and simplifies design by leveraging the product and process knowledge that companies gain from experience and from industry best practices. It includes tools that designers can use to capture knowledge to automated repetitive tasks. The result is reduced cost and cycle time and improved quality.

In product design tolerances are specified due to inherent variabilities of manufacturing processes. Tolerance specifications have significant implications on the quality and cost of the product for proper tolerance specifications, tolerance analysis must be performed. Prototyping is

the only method available for the analysis of the geometric variations.

General Operation

Start with a Sketch

Use the Sketcher to freehand a sketch, and dimension an "outline" of Curves. You can then sweep the sketch using Extruded Body or Revolved Body to create a solid or sheet body. You can later refine the sketch to precisely represent the object of interest by editing the dimensions and by creating relationships between geometric objects. Editing a dimension of the sketch not only modifies the geometry of the sketch, but also the body created from the sketch.

- **Creating and Editing Features**

Feature Modeling lets you create features such as holes, slots and grooves on a model. You can then directly edit the dimensions of the feature and locate the feature by dimensions. For example, a Hole is defined by its diameter and length. You can directly edit all of these parameters by entering new values. You can create solid bodies of any desired design that can later be defined as a form feature using User Defined Features. This lets you create your own custom library of form features.

- **Associativity**

Associatively is a term that is used to indicate geometric relationships between individual portions of a model. These relationships are established as the designer uses various functions for model creation. In an associative model, constraints and relationships are captured automatically as the model is developed. For example, in an associative model, a through hole is associated with the faces that the hole penetrates. If the model is later changed so that one or both of those faces moves, the hole updates automatically due to its association with the faces. See Introduction to Feature Modeling for additional details.

- **Positioning a Feature**

Within Modeling, you can position a feature relative to the geometry on your model using Positioning Methods, where you position dimensions. The feature is then associated with that geometry and will maintain those associations whenever you edit the model. You can also edit the position of the feature by changing the values of the positioning dimensions.

- **Reference Features**

You can create reference features, such as Datum Planes, Datum Axes and Datum CSYS, which you can use as reference geometry when needed, or as construction devices for other features. Any feature created using a reference feature is associated to that reference feature and retains that association during edits to the model. You can use a datum plane as a reference plane in constructing sketches, creating features, and positioning features. You can use a datum axis to create datum planes, to place items concentrically, or to create radial patterns.

- **Expressions**

The Expressions tool lets you incorporate your requirements and design restrictions by defining mathematical relationships between different parts of the design. For example, you can define the height of a boss as three times its diameter, so that when the diameter changes, the height changes also.

- **Boolean Operations**

Modeling provides the following Boolean Operations: Unite, Subtract, and Intersect. Unite combines bodies, for example, uniting two rectangular blocks to form a T-shaped solid body. Subtract removes one body from another, for example, removing a cylinder from a block to form a hole. Intersect creates a solid body from material shared by two solid bodies. These operations can also be used with free form features called sheets.

- **Undo**

You can return a design to a previous state any number of times using the Undo function. You do not have to take a great deal of time making sure each operation is absolutely correct, because a mistake can be easily undone. This freedom to easily change the model lets you cease worrying about getting it wrong, and frees you to explore more possibilities to get it right.

- **Additional Capabilities**

Other Unigraphics NX applications can operate directly on solid objects created within Modeling without any translation of the solid body. For example, you can perform drafting, engineering analysis, and NC machining functions by accessing the appropriate application. Using Modeling, you can design a complete, unambiguous, three dimensional model to describe an object. You can extract a wide range of physical properties from the solid bodies, including mass properties. Shading and hidden line capabilities help you visualize complex assemblies. You can identify interferences automatically, eliminating the need to attempt to do so manually. Hidden edge views can later be generated and placed on drawings. Fully associative dimensioned drawings can be created from solid models using the appropriate options of the Drafting application. If the solid model is edited later, the drawing and dimensions are updated automatically.

- **Parent/Child Relationships**

If a feature depends on another object for its existence, it is a child or dependent of that object. The object, in turn, is a parent of its child feature. For example, if a HOLLOW (1) is created in a BLOCK (0), the block is the parent and the hollow is its child. A parent can have more than one child, and a child can have more than one parent. A feature that is a child can also be a parent of other features. To see all of

the parent-child relationships between the features in your work part, open the Part Navigator.

- One method uses the Edit Feature dialog to change the value of a parameter in each root feature of a part, and then change it back before leaving the Edit Feature dialog. This method produces a genuine net null update if used correctly, but you should ensure that you changed a parameter in every root feature (and that you returned all the parameters to their original values) before you trigger the update.

Introduction to Ansys:

The ANSYS program is self contained general purpose finite element program developed and maintained by Swason Analysis Systems Inc. The program contain many routines, all inter related, and all for main purpose of achieving a solution to an an engineering problem by finite element method.

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

- Build computer models or transfer CAD models of structures, products, components, or systems.
- Apply operating loads or other design performance conditions
- Study physical responses ,such as stress levels, temperature distributions, or electromagnetic fields
- Optimize a design early in the development process to reduce production costs.
- Do prototype testing in environments where it otherwise would be undesirable or impossible

The ANSYS program has a compressive graphical user interface (GUI) that gives users easy, interactive access to program functions, commands, documentation, and reference material. An intuitive menu system helps users navigate through the ANSYS Program. Users can input data using a mouse, a keyboard, or a

combination of both. A graphical user interface is available throughout the program, to guide new users through the learning process and provide more experienced users with multiple windows, pull-down menus, dialog boxes, tool bar and online documentation.

Organization of the Ansys program:

The ANSYS program is organized into two basic levels:

- Begin level
- Processor (or Routine) level

The begin level acts as a gateway in to and out of the ANSYS program. It is also used for certain global program controls such as changing the job name, clearing (zeroing out) the database, and copying binary files. When we first enter the program, we are at the begin level.

At the processor level, several processors are available; each processor is a set of functions that perform a specific analysis task. For example, the general preprocessor (PREP7) is where we build the model, the solution processor(SOLUTION)is where we apply loads and obtain the solution, and the general postprocessor(POST1) is where we evaluate the results and obtain the solution. An additional postprocessor (POST26), enables we to evaluate solution results at specific points in the model as a function of time.

Performing a typical Ansys analysis:

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. The analysis guide manuals in the ANSYS documentation set describe specific procedures for performing analysis for different engineering disciplines.

A typical ANSYS analysis has three distinct steps

- Build the model
- Apply loads and obtain the solution
- Review the results

The following table shows the brief description of steps followed in each phase.

PRE-PROCESSOR	SOLUTION PROCESSOR	POST-PROCESSOR
Assigning element type	Analysis definition	Read results
Geometry definition	Constant definition	Plot results on graphs
Assigning real constants	Load definition	View animated results
Material definition	Solve	
Mesh generation		
Model display		

PRE-PROCESSOR:

The input data for an ANSYS analysis are prepared using a pre-processor. The general pre-processor (PREP 7) contains powerful solid modeling and mesh generation capabilities, and is also used to define all other analysis data with the benefit of data base definition and manipulation of analysis data. Parametric input, user files, macros and extensive online documentation are also available, providing more tools and flexibility. For the analyst to define the problem. Extensive graphics capability is available throughout the ANSYS program, including isometric, perspective, section, edge, and hidden-line displays of three-dimensional structures, graphs of input quantities and results, and contour displays of solution results.

The pre-processor stage involves the following:

- Specify the title, which is the name of the problem. This is optional but very useful, especially if a number of design iterations are to be completed on the same base mode.
- Setting the type of analysis to be used, e.g., Structural, Thermal, Fluid, or electromagnetic, etc
- Creating the model. The model may be created in pre-processor, or it can be imported from another CAD drafting package via a neutral file format.
- Defining element type, these chosen from element library.
- Assigning real constants and material properties like young's modulus, Poisson's ratio, density, thermal

conductivity, damping effect, specific heat, etc

- Apply mesh. Mesh generation is the process of dividing the analysis continuum into number of discrete parts of finite elements.

SOLUTION PROCESSOR

Here we create the environment to the model, i.e., applying constraints & loads. This is the main phase of the analysis, where the problem can be solved by using different solution techniques. Her three major steps involved:

- Solution type required, i.e. static, modal, or transient etc., is selected
- Defining loads. The loads may be point loads, surface loads; thermal loads like temperature, or fluid pressure, velocity are applied.
- Solve FE solver can be logically divided in o three main steps, the pre-solver, the mathematical-engine and post-solver. The pre-solver reads the model created by pre-processor and formulates the mathematical representation of the model and calls the mathematical-engine, which calculates the result. The result return to the solver and the post solver is used to calculate strains, stresses, etc., for each node within the component or continuum.

POST –PROCESSOR:

Post processing means the results of an analysis. It is probably the most important step in the analysis, because we are trying to understand how the applied loads affects the design, how food your finite element mesh is, and so on.

The analysis results are reviewed using postprocessors, which have the ability to display distorted geometries, stress and strain contours, flow fields, safety factor contours, contours of potential field results; vector field displays mode shapes and time history graphs. The postprocessor can also be used for algebraic operations, database manipulators, differentiation, and integration of calculated results. Response

spectra may be generated from dynamic analysis. Results from various loading may be harmonically loaded axis metric structures.

LITERATURE REVIEW

1. Experimental Test on Carbon Fiber/Epoxy and Glass Fiber /Epoxy Pultruded Rods for Mechanical Properties by Krishna et al. This project explains, composite rods of glass fiber/epoxy and carbon fiber/epoxy with 80% of fiber loading and 20% of matrix are produced by using Pultrusion technique. Pultrusion is a continuous process for manufacture of composite materials with constant cross-section. Reinforced fibers are pulled through a resin, possibly followed by a separate performing system, and into a heated die, where the resin undergoes polymerization. Many resin types may be used in Pultrusion including polyester, polyurethane, vinyl ester and epoxy. These two types of rods are subjected to tensile test, chemical absorption and impact tests at various temperatures like room temperature and at below room temperatures.

PROBLEM STATEMENT & METHODOLOGY

Problem statement

In this project we designed ladder stool with pultrusion rods which are made by pultrusion method. Ladder stool forms the stresses under loading condition. So if we not select proper material for manufacturing of stool it may causes fatigue failures.

Methodology

1. At first, Both Glass fiber/Epoxy rods and Carbon fiber/Epoxy rods are manufactured by Pultrusion method.
2. 3d modeling of Pultruded rods is done by using NX software.
3. The 3d model of Pultruded rods is converted to parasolid file.
4. Then, the parasolid file is imported to ANSYS software to perform finite element analysis of Pultruded rods.
5. Static analysis is done on Pultruded rods to determine the deflections and stresses for static loads.

6. Static analysis of Pultruded rods is carried out for different materials (i.e. Carbon/Epoxy, Glass/Epoxy materials.).

7. Comparison of results is done to determine the better material.

MANUFACTURING OF CARBON FIBER/EPOXY RODS AND GLASS FIBER/EPOXY RODS BY PULTRUSION METHOD

4.1PULTRUSION METHOD:

Pultrusion is the process of “pulling” raw composites through a heated die to create a continuous composite profile. It is a continuous process for manufacture of composite materials with constant cross-section. The term Pultrusion combines the words, “pull” and “extrusion”. Extrusion is the pushing of material, such as a billet of aluminum, through a shaped die. Pultrusion is the pulling of material, such as carbon fiber and resin, through a shaped die.

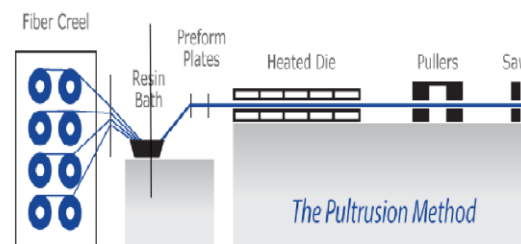


Fig shows Pultrusion method

STEPS INVOLVED IN PULTRUSION METHOD:

- Fiber creel
- Resin bath
- Perform plates
- Heated die
- Hydraulic pullers
- Saw

The typical Pultrusion process starts with racks or creels holding spools of bundled continuous fiber (roving) most often the reinforcement is fiberglass, but it can be carbon, aramid, or a mixture. This raw fiber is pulled off the racks and guided through a resin bath or resin impregnation system. The raw resin is a thermosetting resin and combined with fillers, catalyst and pigments. The un-cured composite material is guided through a series of tooling. This

custom tooling helps arrange and organize the fiber into the correct shape, while excess resin is squeezed out, also known as “Debuling”. This tooling is known as “pre-former”. Often continuous strand mat and surface veils are added in this step to increase structure and surface finish. Once the resin impregnated fiber is organized and removed of excess resin, the composite will pass through a heated steel die. Precisely machined and often chromed, the die is heated to a constant temperature, and may have several zones of temperature throughout its length, which will cure the thermosetting resin. The profile that exits the die is now a cured Pultruded Fiber Reinforced Plastic (FRP) composite. This FRP profile is pinched and pulled by a “gripper” system. Either caterpillar tracks or hydraulic clamps are used to pull the composite through the Pultrusion die on a continuous basis. At the end of this Pultrusion machine is a cut-off saw. The Pultruded profiles are cut to the specific length and stacked for delivery.

MANUFACTURING OF GLASS FIBRE/EPOXY:

The raw material used for manufacturing of rods is

- Glass fiber roving
- Epoxy resin

Glass fiber roving:

Fiber is the reinforcing phase of a composite. Roving is made up of fiber glass unidirectional filaments, which are manufactured in continuous rolls. Roving is always preset in Pultruded products comprising 50% to 70% of the total glass content.

Properties of glass fiber roving:

Properties	values
Tensile strength(MPa)	2200-2800
Young's modulus (GPa)	80-90
Density(g/cm ³)	2.50-2.57

Epoxy resin:

Epoxy resin is taken as the matrix material. An epoxy resin is defined as a

molecule with more than one epoxy group, which can be hardened into a usable plastic.

Manufacturing process:

The rods were manufactured by using Pultrusion process. The Pultrusion process consists of

- Fiber creel
- Resin bath
- Perform plates
- Heated die
- Hydraulic pullers
- Saw

Manufacturing of Composite Rods:

The two different types of reinforcement were used to prepare composite rods they are: glass fiber roving (4800 Tex) collected from Advanced Millennium Materials, Cherlapally. The matrix material used for manufacturing of Composite rods is epoxy resin with L12 Grade. For this epoxy system, Hardener K-12 Grade, Accelerator K-13 Grade was supplied by Atul Ltd and internal mold releasing agent INT-1846-N2 was used.

Totally 3kgs of matrix mixture is used for single rod production. The ratio of resin to hardener is 1:1 (i.e. 1.5kgs of epoxy resin and 1.5kgs of hardener), 20ml of accelerator and 15ml of mould releasing agent were mixed in the resin mixture. Stir the mixture gently with the stirrer in order to avoid air bubbles.

Rods with uniform diameter of 22 mm were produced by using pultrusion method. For Glass/epoxy rod 110 spools of glass rovings are used for producing required diameter and the percentage of fiber loading is 85% and the percentage of matrix is 15%.

Extrusion is pulling of material such as Fiber glass and Resin, through a shaped and heated die. Thermocouples were mounted on the die. The Process begins with the fibers coming from the Rovings. Then there is wetting by resin called as Resin Impregnation. Impregnation fibers are passed through heated die through heated dies polymerizing and curing takes place at 165 degree centigrade. The pulling speed of

the hydraulic pullers is 6 meters/hr. Finally, Eglassfibre /Epoxy rod is manufactured.

MANUFACTURING OF CARBON FIBRE/EPOXY:

The raw material used for manufacturing of rods is

- Carbon fiber roving
- Epoxy resin

Carbon fiber roving:

Fiber is the reinforcing phase of a composite. Roving is made up of fiber Carbon unidirectional filaments, which are manufactured in continuous rolls. Roving is always preset in Pultruded products comprising 50% to 70% of the total Carbon content.

Properties of Carbon fiber roving:

Properties	values
Tensile strength(MPa)	1100-1900
Young's modulus (GPa)	120-140
Density(g/cm ³)	1.0-1.8

Epoxy resin:

Epoxy resin is taken as the matrix material. An epoxy resin is defined as a molecule with more than one epoxy group, which can be hardened into a usable plastic.

Manufacturing process:

The rods were manufactured by using Pultrusion process.

The Pultrusion process consists of

- Fiber creel
- Resin bath
- Perform plates
- Heated die
- Hydraulic pullers
- Saw

Manufacturing of Composite Rods:

The two different types of reinforcement were used to prepare composite rods they are: carbon fiber roving (6K) collected from Advanced Millennium Materials, Cherlapally. The matrix material used for manufacturing of Composite rods is epoxy resin with L12 Grade. For this epoxy system, Hardener K-12 Grade, Accelerator K-13 Grade was supplied by Atul Ltd and internal mold releasing agent INT-1846-N2 was used.

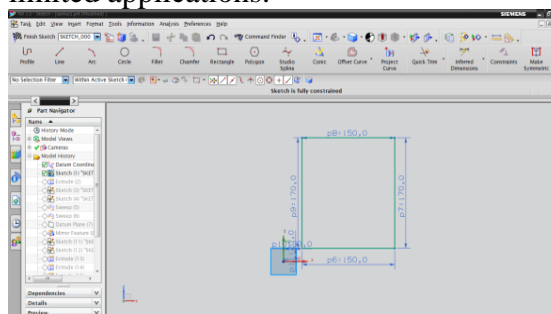
Totally 4kgs of matrix mixture is used for single rod production. The ratio of resin to hardener is 1:1 (i.e. 2kgs of epoxy resin and 2kgs of hardener), 80ml of accelerator and 40ml of mold releasing agent were mixed in the resin mixture. Stir the mixture gently with the stirrer in order to avoid air bubbles.

Rods with uniform diameter of 22 mm were produced by using pultrusion method. For Carbon/epoxy rod 820 spools of Carbonrovings are used for producing required diameter and the percentage of fiber loading is 85% and the percentage of matrix is 15%.

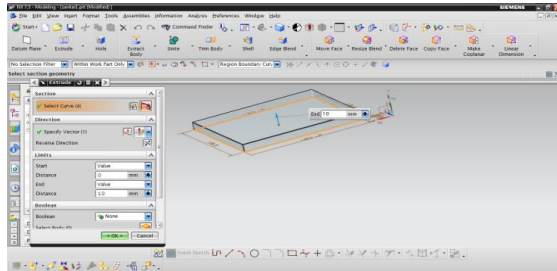
Extrusion is pulling of material such as Fiber Carbon and Resin, through a shaped and heated die. Thermocouples were mounted on the die. The Process begins with the fibers coming from the Rovings. Then there is wetting by resin called as Resin Impregnation. Impregnation fibers are passed through heated die through heated dies polymerizing and curing takes place at 165 degree centigrade. The pulling speed of the hydraulic pullers is 6 meters/hr. Finally, a Carbon fibre /Epoxy rod is manufactured.

3D MODELLING OF COMPOSITE PULTRUDED RODS

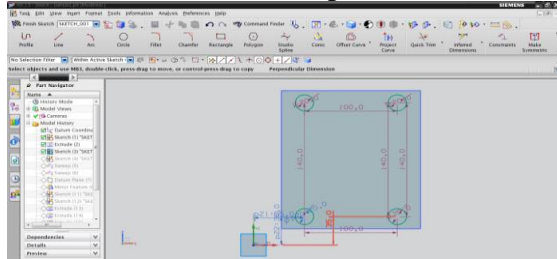
3d modeling of Pultruded rods were generated in Unigraphics software. Unigraphics software is one the cad software mostly used in India now a day. It is mostly used in small industries due to its limited applications.



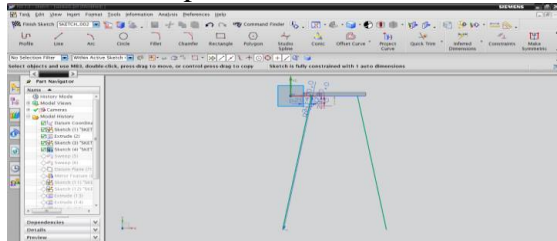
2D Sketch of pultrusion rod



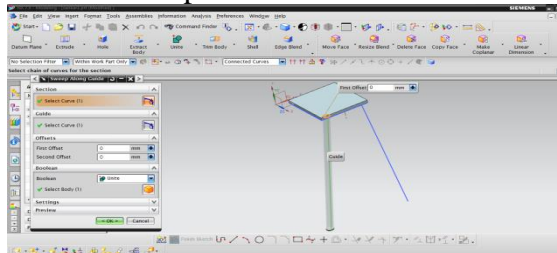
Extrude of 2D Sketch of pultrusion rod



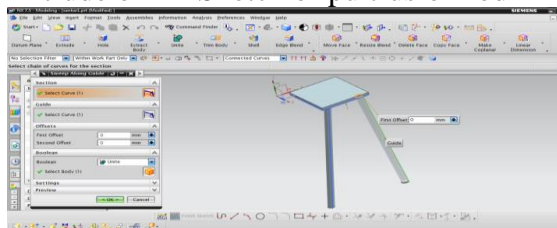
2D Sketch of pultrusion rod



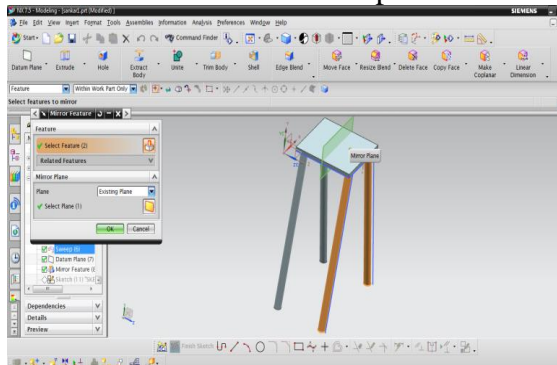
2D Sketch of pultrusion rod



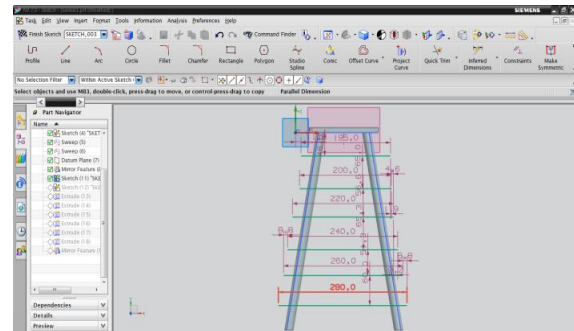
Extrude of 2D Sketch of pultrusion rod



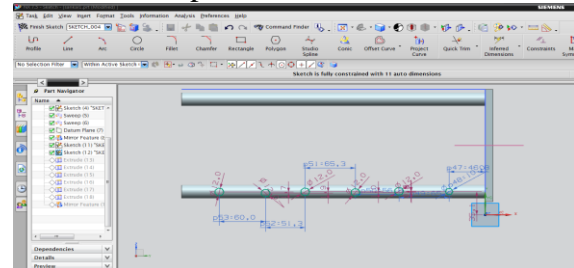
5.6 Extrude of 2D Sketch of pultrusion rod



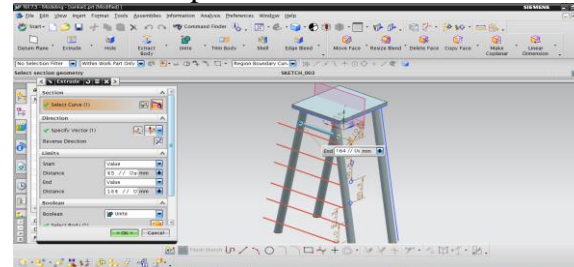
5.7 Extrude of 2D Sketch of pultrusion rod



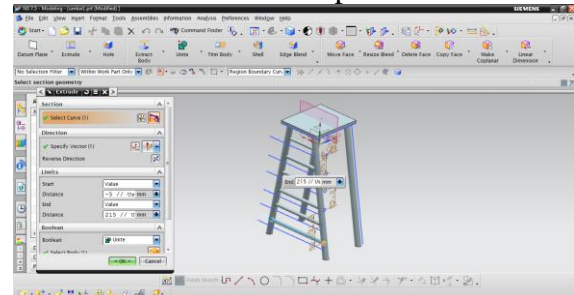
2D Sketch of pultrusion rod



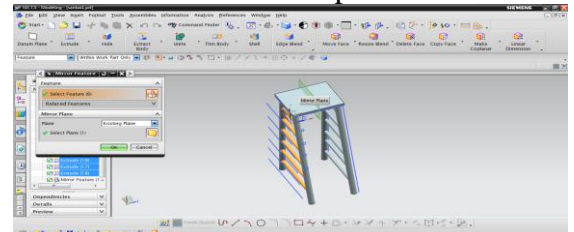
2D Sketch of pultrusion rod



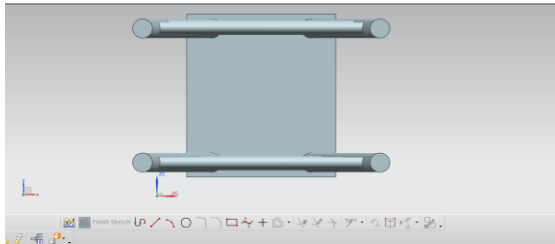
Extrude of 2D Sketch of pultrusion rod



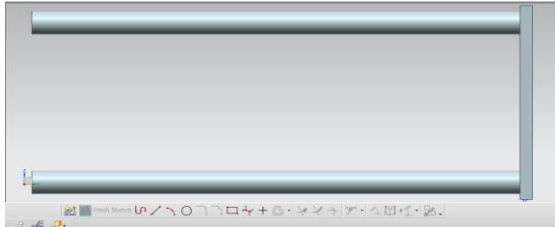
Extrude of 2D Sketch of pultrusion rod



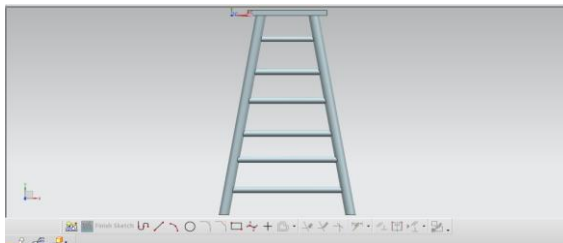
Extrude of 2D Sketch of pultrusion rod



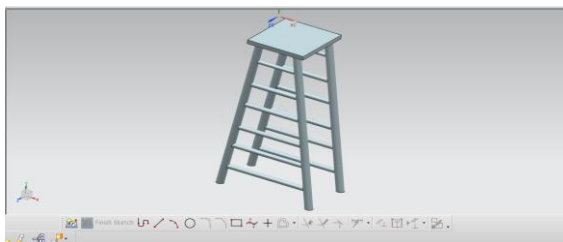
Shows Top view of Pultruded rod



shows Side view of Pultruded rod



shows front view of Pultruded rod



shows Isometric view of Pultruded rod

STRUCTURAL ANALYSIS OF LADDER OF PULTRUDED RODS FOR EGLASS FIBRE/EPOXY MATERIAL

Structural analysis comprises the set of physical laws and mathematics required to study and predicts the behavior of structures. The subjects of structural analysis are engineering artifacts whose integrity is judged largely based upon their ability to withstand loads; they commonly include buildings, bridges, aircraft, and ships. Structural analysis incorporates the fields of mechanics and dynamics as well as the many failure theories. From a theoretical perspective the primary goal of structural analysis is the computation of deformations, internal forces, and stresses. In practice, structural analysis can be viewed more abstractly as a method to drive the

engineering design process or prove the soundness of a design without a dependence on directly testing it.

METHODS OF PERFORMING STRUCTURAL ANALYSIS

To perform an accurate analysis a structural engineer must determine such information as structural loads, geometry, support conditions, and materials properties. The results of such an analysis typically include support reactions, stresses and displacements. This information is then compared to criteria that indicate the conditions of failure. Advanced structural analysis may examine dynamic response, stability and non-linear behavior.

DESCRIPTION:

ANSYS is a Finite Element Analysis (FEA) code widely used in the Computer Aided Engineering (CAE) field. ANSYS software allow to construct computer models of structures, machine components or systems, apply operating loads and other design criteria and study physical responses, such as stress levels, temperature distributions, pressure, etc. The ANSYS program has a variety of design analysis applications, ranging from automobiles to such highly sophisticated systems as aircraft, nuclear reactor containment buildings and bridges. There are 250+ elements derived for various applications in ANSYS. In the present application shell, beam and mass elements that have structural static and dynamic analysis capabilities were considered.

FINITE ELEMENT MODELING:

3D model of the Ladder was developed in PRO-E. The model was then converted into aIGES to import into ANSYS. A Finite Element model was developed with solid elements. The elements that are used for idealizing the Ladder were described below. A detailed Finite Element model was built with solid elements to idealize all the components of the Ladder. The elements that are used for idealizing the Ladder are solid92. The description of each element is given below.

MATERIAL PROPERTIES:

Considering Ladder is made of Eglass/Epoxy.

Properties of Eglass/Epoxy material:

Table. Properties of E-GLASS/Epoxy

S. No	Property	Units	E-Glass/Epoxy
1.	E_{11}	GPa	50.0
2.	E_{22}	GPa	12.0
3.	G_{12}	GPa	5.6
4.	ν_{12}	-	0.3
5.	$S^t_1 = S^c_1$	MPa	800.0
6.	$S^t_2 = S^c_2$	MPa	40.0
7.	S_{12}	MPa	72.0
8.	ρ	Kg/m ³	2000.0

Yield strength = 800MPa

CASE-1:STRUCTURAL ANALYSIS IS CARRIED BY APPLYING FORCE ON THE TOP SURFACE OF THE LADDER



Fig shows infinite model of Ladder

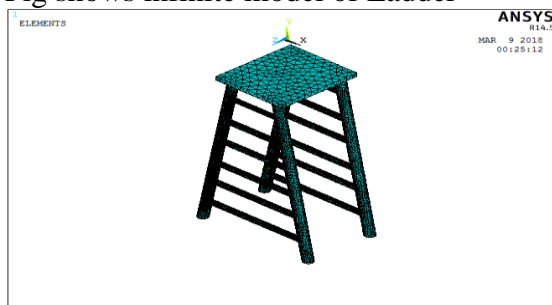


Fig shows finite model of Ladder

Boundary conditions:

- A load of 1000N is applied on top surface of the ladder.
- Bottom of Ladder is constrained in all dof

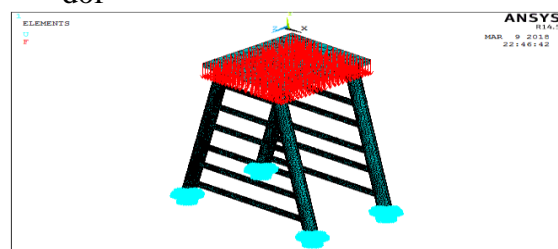


Fig shows boundary and load conditions on Ladder

RESULTS:

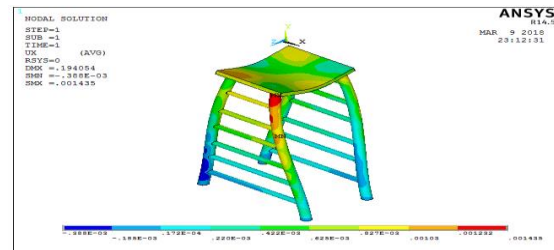


Fig shows displacement on the Ladder in X-Direction

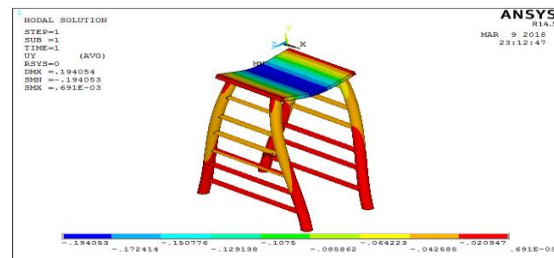


Fig shows displacement on the Ladder in Y-Direction

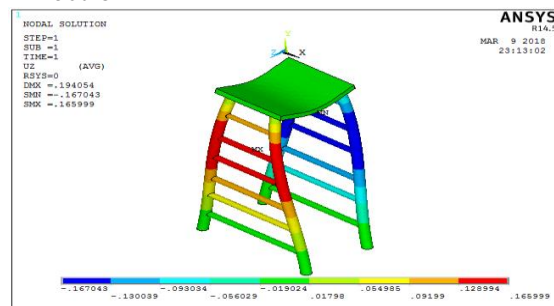


Fig shows displacement on the Ladder in Z-Direction

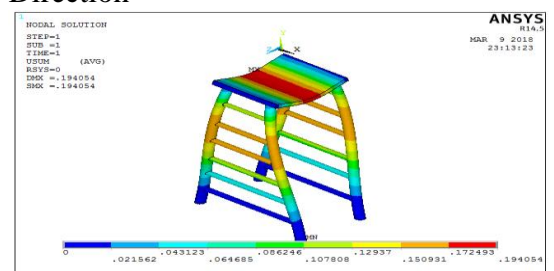


Fig shows Total displacement on the Ladder

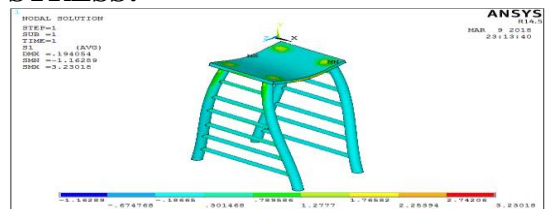


Fig shows 1st principal stress observed on the Ladder

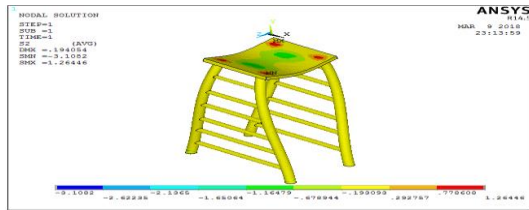


Fig shows 2nd principal stress observed on the Ladder

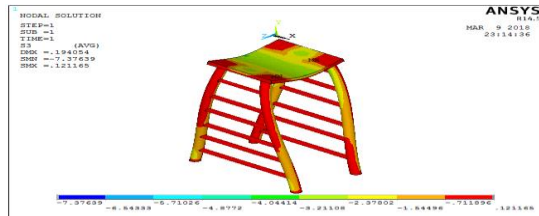


Fig shows 3rd principal stress observed on the Ladder

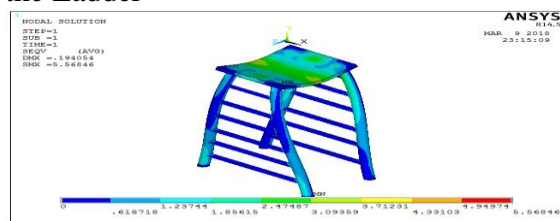


Fig shows Von misses stress observed on the Ladder

SLNo	Material	Displacement(mm)	Stress(MPa)	Factor of safety
1	Glass/Epoxy	0.194	5.56	143.88

From analysis results, the resultant displacement observed on Ladder is **0.194 mm**. The Vonmisses stress observed on Ladder is **5.56MPa**. The yield strength of Carbon/Epoxy material is **800MPa**. The Vonmisses stress of Ladder is less than the yield strength of the material. Hence the Ladder is a safe in design for Carbon/Epoxy material with a factor of safety(yield stress/working stress) of 143.88.

CASE-2:STRUCTURAL ANALYSIS IS CARRIED BY APPLYING THE LOAD ON THE LEG OF THE LADDER:

Boundary conditions:

- A load of 1000N is applied on the leg of the ladder.
- Bottom of Ladder is constrained in all dof.

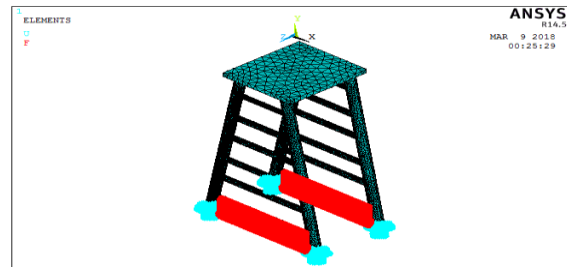


Fig shows boundary and load conditions on Ladder
RESULTS:

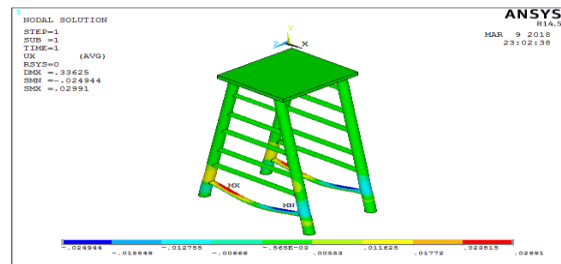


Fig shows displacement on the Ladder in X-Direction

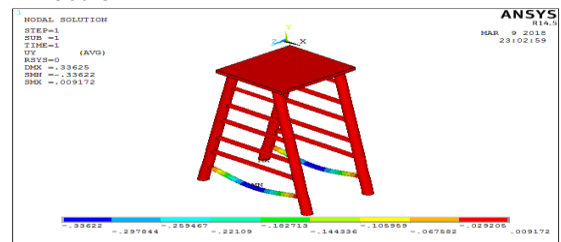


Fig shows displacement on the Ladder in Y-Direction

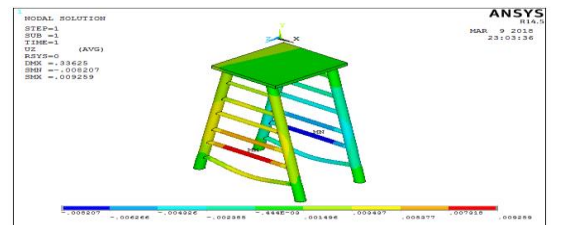


Fig shows displacement on the Ladder in Z-Direction

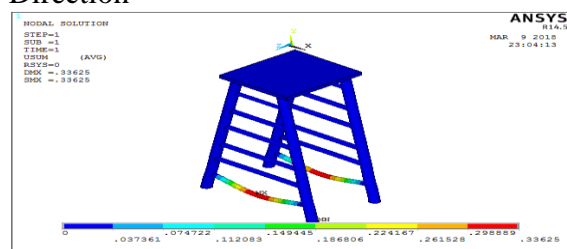


Fig shows Total displacement on the Ladder

STRESS:

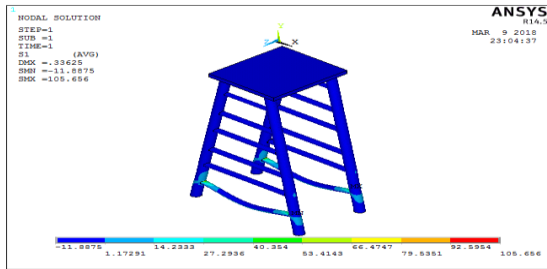


Fig shows 1st principal stress observed on the Ladder

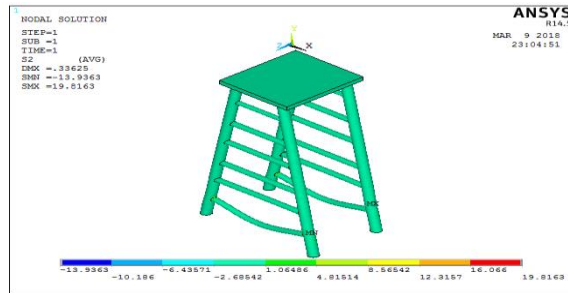


Fig shows 2nd principal stress observed on the Ladder

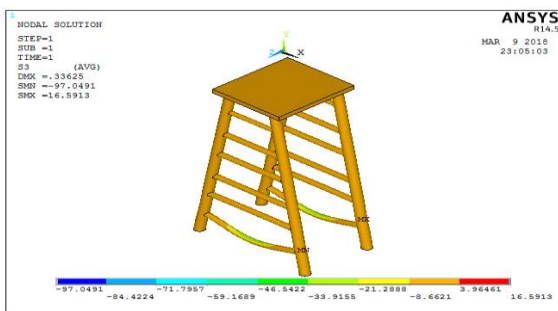


Fig shows 3rd principal stress observed on the Ladder

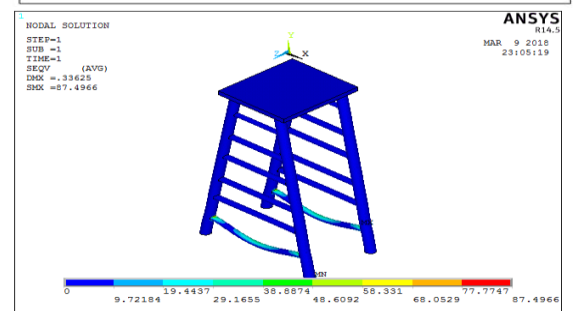
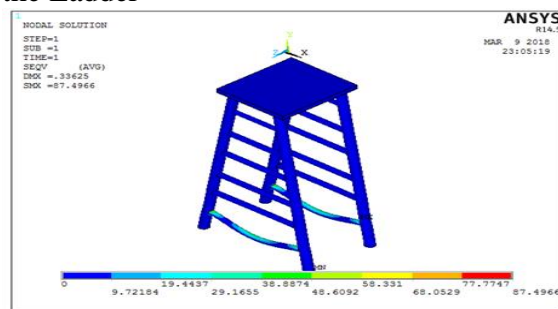


Fig shows Von misses stress observed on the Ladder

SLNo	Material	Displacement	Stress	Factor of safety
1	Eglass/Epoxy	0.33mm	87.49MPa	9.143

From analysis results, the resultant displacement observed on Ladder is **0.33mm**. The Vonmises stress observed on Ladder is **87.49MPa**. The yield strength of Eglass/Epoxy material is **800MPa**. The Vonmises stress of Ladder is less than the yield strength of the material. Hence the Ladder is a safe in design for Glass/Epoxy material with a factor of safety of 9.143.

STRUCTURAL ANALYSIS OF LADDER OF PULTRUDED RODS FOR CARBON FIBRE/EPOXY MATERIAL

Structural analysis comprises the set of physical laws and mathematics required to study and predicts the behavior of structures. The subjects of structural analysis are engineering artifacts whose integrity is judged largely based upon their ability to withstand loads; they commonly include buildings, bridges, aircraft, and ships. Structural analysis incorporates the fields of mechanics and dynamics as well as the many failure theories. From a theoretical perspective the primary goal of structural analysis is the computation of deformations, internal forces, and stresses. In practice, structural analysis can be viewed more abstractly as a method to drive the engineering design process or prove the soundness of a design without a dependence on directly testing it.

METHODS OF PERFORMING STRUCTURAL ANALYSIS

To perform an accurate analysis a structural engineer must determine such information as structural loads, geometry, support conditions, and materials properties. The results of such an analysis typically include support reactions, stresses and displacements. This information is then compared to criteria that indicate the conditions of failure. Advanced structural analysis may examine dynamic response, stability and non-linear behavior.

DESCRIPTION:

ANSYS is a Finite Element Analysis (FEA) code widely used in the Computer Aided Engineering (CAE) field. ANSYS software allow to construct computer models of structures, machine components or systems, apply operating loads and other design criteria and study physical responses, such as stress levels, temperature distributions, pressure, etc. The ANSYS program has a variety of design analysis applications, ranging from automobiles to such highly sophisticated systems as aircraft, nuclear reactor containment buildings and bridges. There are 250+ elements derived for various applications in ANSYS. In the present application shell, beam and mass elements that have structural static and dynamic analysis capabilities were considered.

FINITE ELEMENT MODELING:

3D model of the Ladder was developed in PRO-E. The model was then converted into IGES to import into ANSYS. A Finite Element model was developed with solid elements. The elements that are used for idealizing the Ladder were described below. A detailed Finite Element model was built with solid elements to idealize all the components of the Ladder. The elements that are used for idealizing the Ladder are solid92. The description of each element is given below.

MATERIAL PROPERTIES:

Considering Ladder is made of Carbon/Epoxy.

Properties of Carbon/Epoxy material:

Table. Properties of HM Carbon/Epoxy

S. No	Property	Units	HM Carbon/Epoxy
1.	E_{11}	GPa	190.0
2.	E_{22}	GPa	7.7
3.	G_{12}	GPa	4.2
4.	ν_{12}	-	0.3
5.	$S^*_1 = S^*_c_1$	MPa	870.0
6.	$S^*_2 = S^*_c_2$	MPa	94.0
7.	S_{12}	MPa	30.0
8.	ρ	Kg/m ³	1600.0

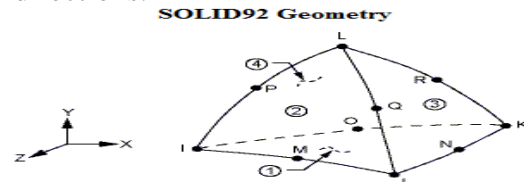
Yield strength = 880 MPa

Solid 92 element description:

10 node solid92:

SOLID92 has quadratic displacement behaviour and is well suited to model irregular meshes (such as produced from

various CAD/CAM systems). See SOLID95 for a 20-node brick shaped element. The element is defined by ten nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.



CASE-1: STRUCTURAL ANALYSIS IS CARRIED BY APPLYING FORCE ON THE TOP SURFACE OF THE LADDER



Fig shows infinite model of Ladder

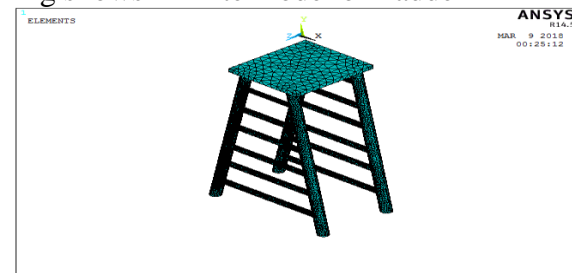


Fig shows finite model of Ladder

Boundary conditions:

- A load of 1000N is applied on top surface of the ladder.
- Bottom of Ladder is constrained in all dof

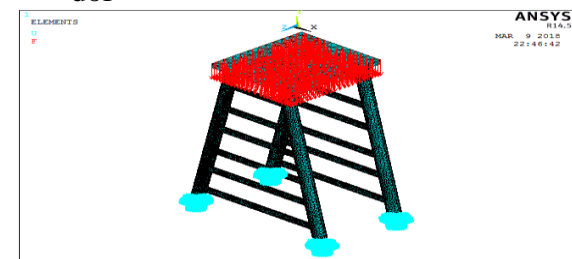


Fig shows boundary and load conditions on Ladder

RESULTS:

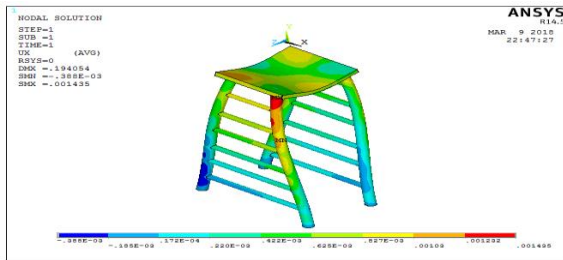


Fig shows displacement on the Ladder in X-Direction

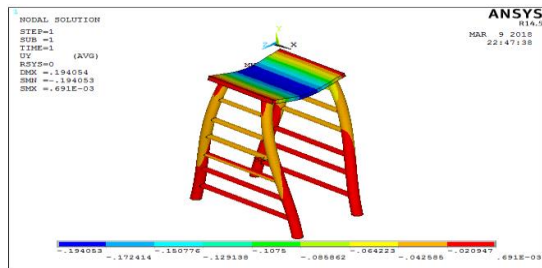


Fig shows displacement on the Ladder in Y-Direction

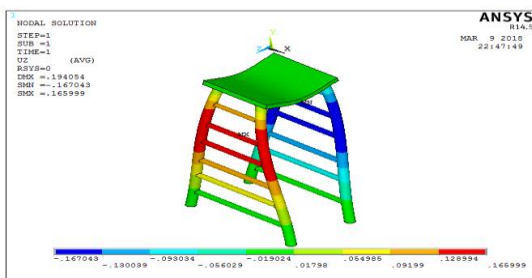


Fig shows displacement on the Ladder in Z-Direction

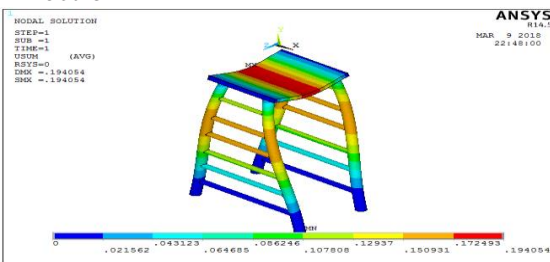


Fig shows Total displacement on the Ladder

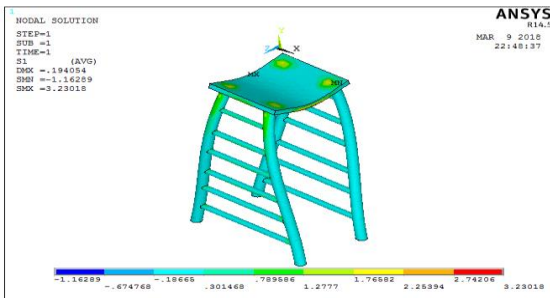


Fig shows 1st principal stress observed on the Ladder

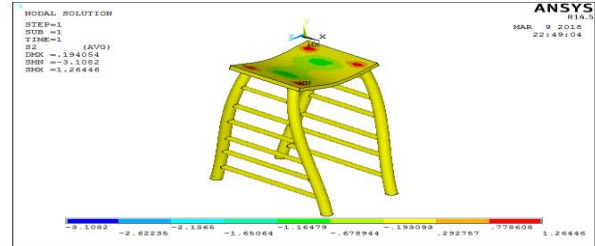


Fig shows 2nd principal stress observed on the Ladder

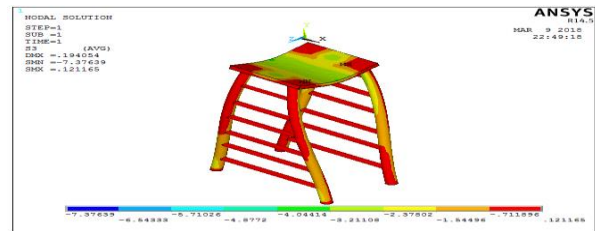


Fig shows 3rd principal stress observed on the Ladder

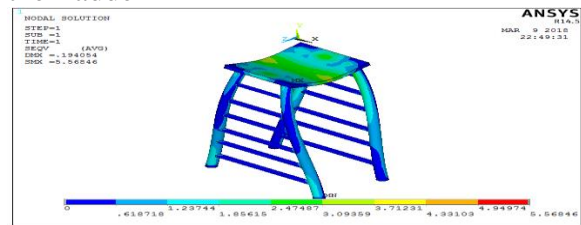


Fig shows Von misses stress observed on the Ladder

Sl.No	Material	Displacement(mm)	Stress(MPa)	Factor of safety
1	Carbon/Epoxy	0.194	5.56	158.27

From analysis results, the resultant displacement observed on Ladder is **0.194 mm**. The Vonmisses stress observed on Ladder is **5.56MPa**. The yield strength of Carbon/Epoxy material is **880MPa**. The Vonmisses stress of Ladder is less than the yield strength of the material. Hence the Ladder is a safe in design for Carbon/Epoxy material with a factor of safety(yield stress/working stress) of 158.27.

CASE-2:STRUCTURAL ANALYSIS IS CARRIED BY APPLYING THE LOAD ON THE LEG OF THE LADDER:

Boundary conditions:

- A load of 1000N is applied on the leg of the ladder.
- Bottom of Ladder is constrained in all dof.

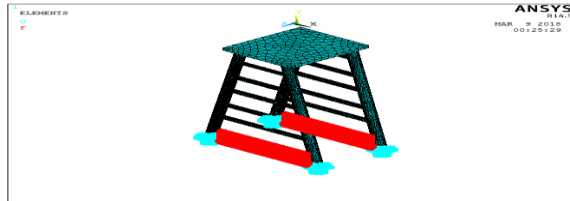


Fig shows boundary and load conditions on Ladder

RESULTS:

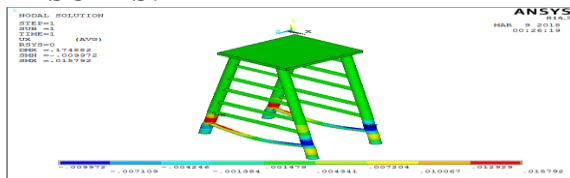


Fig shows displacement on the Ladder in X-Direction

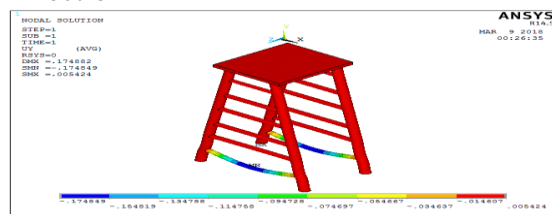


Fig shows displacement on the Ladder in Y-Direction

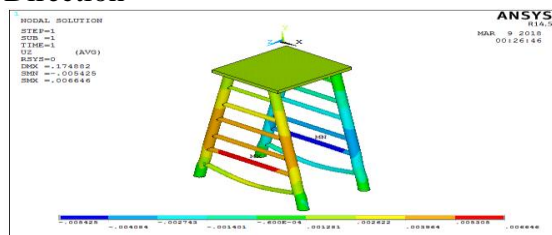


Fig shows displacement on the Ladder in Z-Direction

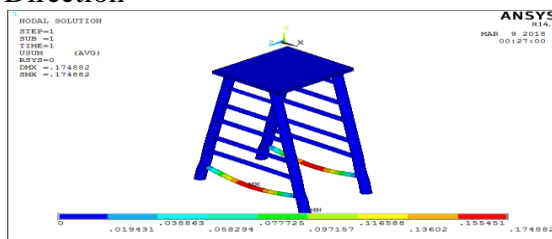


Fig shows Total displacement on the Ladder

STRESS:

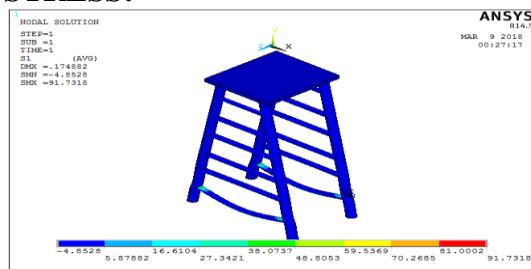


Fig shows 1st principal stress observed on the Ladder

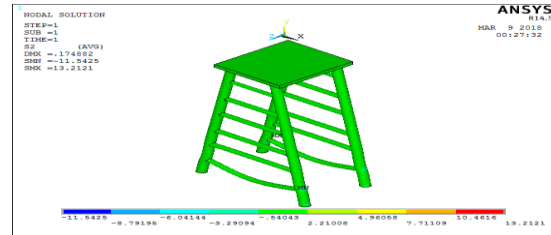


Fig shows 2nd principal stress observed on the Ladder

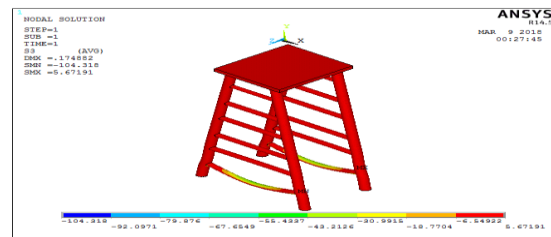


Fig shows 3rd principal stress observed on the Ladder

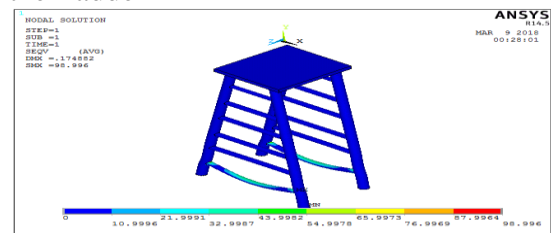


Fig shows Von misses stress observed on the Ladder

Sl.No	Material	Displacement(mm)	Stress(MPa)	Factor of safety
1	Carbon/Epoxy	0.174	98.99	8.88

From analysis results, the resultant displacement observed on Ladder is 0.174 mm. The Vonmisses stress observed on Ladder is 98.99MPa. The yield strength of Carbon/Epoxy material is 880MPa. The Vonmisses stress of Ladder is less than the yield strength of the material. Hence the Ladder is a safe in design for Carbon/Epoxy material with a factor of safety of 8.88.

RESULTS AND CONCLUSION FOR TWO MATERIALS

Structural static analysis was done on the ladder for 2 cases for three materials (i.e. Eglass/Epoxy and Carbon/Epoxy materials).

Case-1: Applying the force on the top surface of the ladder.

Case-2: Applying the force on the leg of the ladder.

1. Results of ladder for Eglass/Epoxy material:

Case-1:Applying the force on the top surface of the ladder for Eglass/Epoxy material:

SL.No	Material	Displacement	Stress	Factor of Safety
1	Eglass/Epoxy	0.29	18.02	44.3

Case-2: Applying the force on the leg of the ladder for Eglass/Epoxy material:

SL.No	Material	Displacement	Stress	Factor of safety
1	Eglass/Epoxy	0.69	21.7	36.8

From the above tables, it was observed that the displacement and stress of the ladder for the case-1 (applying the force on the top surface) is less than the case-2 (applying the force on the leg).

Results of ladder for Carbon/Epoxy material:

Case-1:Applying the force on the top surface of the ladder for Carbon/Epoxy material:

SL.No	Material	Displacement	Stress	Factor of safety
1	Carbon/Epoxy	0.47	14.09	62.4

Case-2: Applying the force on the leg of the ladder for Carbon/Epoxy material:

SL.No	Material	Displacement	Stress	Factor of safety
1	Carbon/Epoxy	0.97	25.64	34.3

	Material	Displacement	Stress	Factor of safety
1	Carbon/Epoxy	0.97	25.64	34.3

From the above tables, it was observed that the displacement and stress of the ladder for the case-1 (applying the force on the top surface) is less than the case-2 (applying the force on the leg).

Comparison of case-1(applying the force on the top surface) of the ladder for two materials:

SL.No	Material	Displacement	Von misses stress	Factor of Safety
1	Eglass/Epoxy	0.29	18.02	44.3
2	Carbon/Epoxy	0.47	14.09	62.4

Comparison of case-2(applying the force on the leg) of the ladder for two materials:

SL.No	Material	Displacement	Von misses stress	Factor of Safety
1	Eglass/Epoxy	0.69	21.7	36.8
2	Carbon/Epoxy	0.97	25.64	34.3

Conclusion:

Both Glass fiber/Epoxy rods and Carbon fiber/Epoxy rods were manufactured by Pultrusion method. The Carbon/Epoxy and Glass/Epoxy rods were subjected to tensile test and Impact test. Then, the 3d modeling of Pultruded rods was done by using Unigraphics software. Static analysis was carried to determine the deflections and stresses for static loads for different materials (i.e. Carbon/Epoxy, Glass/Epoxy materials.). Analysis was done for two cases (i.e. in first case, the force was applied on the top surface of the ladder and in second case, the force was applied on the leg of the ladder.). For First case, Carbon/Epoxy material has less Von misses stress and more Factor of safety than the other material. For second case, E-glass/Epoxy material has high Factor of safety than the other material. So, it was concluded that the Ladder with Carbon/Epoxy Pultruded rods is better compare to other materials for case-1(the force was applied on the top surface of the ladder) and the Ladder with Eglass/Epoxy Pultruded rods is better than other materials for case-2.

REFERENCES

1. Barbero, E.J. Pultruded structural shapes: from the constituents to the structural behavior. SAMPE J. 1991, 27(1), 25
2. Luciano, R. and Barbero, E.J. Formulae for the stiffness of composites with periodic microstructure. Int. J. Solids Struct. 1994, 31(21), 2933
3. Jones, R.M. 'Mechanics of composite materials', Hemisphere Publishing Corporation, New York, NY, 1975
4. Barbero, E.J., Lopez-Anido, R. and Davalos, J.F. On the mechanics of thin-walled laminated composite beams. J. Compos. Mater. 1993, 27(8), 806-983
5. Chamis, C.C. Simplified composites micromechanics equations