Validation and Optimization of Stress Induced using Ansys

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ABSTRACT

-It is difficult to determine the mechanical properties of composite materials like stress,strain etc due to time consumption and extra expenditure in proper mechanical testing and designing.So once methodology is established by using software Ansys , that process will be less time consuming and of less expenditure. Methodology will be established by validating software analysis with the industrial data and then optimized using various resin.

Keywords
Ansys 12.1 , FEM, Static analysis, Dynamic analysis.

1. INTRODUCTION

In todays modern world vehicles are mostly demanded things in each human beings life. For designing any vehicle various parts are required. These parts should be maximum load carrying and they should have maximum life. Si for doing all this we require a lot of man power and money power. So that can be minimized directly by using FEM method on software Ansys thus reducing time and man power in mechanical testing and designing. In this paper propeller shaft of steel is designed and modelled on solidwork software. Then stress which is induced on that is analysed on Ansys software and then it is compared with actual values obtained from industry. Thus validating the analysis and then that is optimized using different resins such as Epoxy resin, Kevlar etc.

2. SOLIDWORKS

SOLIDWORKS is a computer graphics system for modeling various mechanical designs for performing related design and manufacturing operations. The system uses a 3D solid modeling system as the core, and applies the feature base parametric modeling method. In short solidworks is a feature based parametric solid modeling system with many extended design and manufacturing applications.

Difference between solidworks and other CAD systems:

Solidworks is the first commercial CAD system entirely based upon the feature based design and parametric modeling philosophy. Today many software producers have recognized the advantage of this approach and started to shift their product on to this platform. Nevertheless, the differences between a feature based, parametric solid modeling.

Solidworks	**Conventional CAD systems**
Solid model	Wire frame and solid model
Parametric model	Fixed-dimension model
Feature based modeling	Primitive-based modeling
Subject oriented sub-modeling systems	A single geometry based system

The following are some important features of **SOLIDWORKS:**

- **Ease of use:**

Solidworks was designed to begin where the design engineers begins with features and design criteria. Solidworks menus flow in the manner that is easily understand. This makes it simple to learn and utilize even for the most casual user. Because Solidworks provides the ability to sketch directly on the solid model, feature placement is simple and accurate.

- Full associatively:

Solidworks is based on a single data structure with the ability to make change built into the system. Therefore, when a change is made anywhere in the development process it is propagated throughout the entire design through manufacturing process, ensuring consistency in all engineering deliverables.

- Parametric & feature based modeling:

Solidworks features are process plans with embedded intelligence and are easy to use while at the same time powerful enough to most complex geometry.

- Powerful assembly capabilities:

Assembling components is easy with Solidworks simply tell the system to "mate", "insert", or "align" the components and they are assembled, always maintaining the design intent. Also, the components "know" how they are related, so if one changes, either position ally or geometrically, the other will change accordingly. Parts can be designed right in the assembly and defined by other components, so if they move or change size, the part will automatically update to reflect the change.

- Robustness:

This provides the engineer with the most accurate representation, of geometry, mass properties, and interference checking available.

- Change management:

Powerful change capabilities are inherent with Solidworks full associatively, enabling design through manufacturing disciplines to execute their functions in parallel.

- Hardware independence:

Solidworks runs on all of the major UNIX and Windows NT platforms, maintaining the same look and feel on every system. Users can select the most economical hardware configuration for their needs, and mix and match any combination of platform.

Functions of solidworks

- Part design:
  - Create sketch of the parts.

- Sketch cosmetic features,
- Create geometric tolerances and surface finishes on models.
- Assign the properties like density, mass, units, materials etc.

- Assembly design:
  - Create full product assembly.
  - Disassemble components from assembly.
  - Modify part dimensions.
  - Additional functionality is also available,
- Design documentation (drawings):
  - Create detailed, exploded, auxiliary, cross-sectional prospective drawing views,
  - Perform extensive view modifications.
  - Modify dimension values and number of digits.
  - Include existing geometric tolerances.
- General functionality:
  - Database management commands.
  - Larger control of placing items.
  - Measurement commands.
  - Viewing capabilities.

Functions of modules of SOLIDWORKS

The core of Solidworks is the feature-based, parametric solid modeling system for modeling mechanical parts. The part model created by this system can be used to form mechanical assemblies and to produce engineering drawings. The model can also be used lo carry out other related manufacturing activities such as the generation of CNC tool paths and Bills of Material. These extended functions are reflected by the following Solidworks modes:

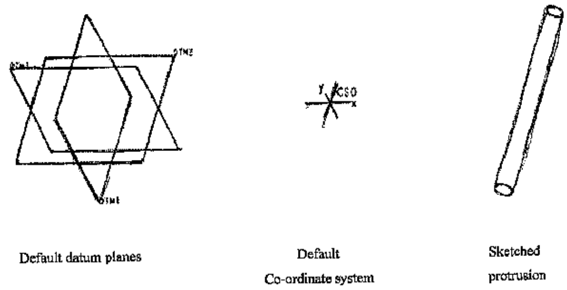
Table 1. Functional modules of SOLIDWORKS

Mode	Description
Sketcher	Sketch feature sections and parametric drawings
Assembly	Form the solid model of an assembly
Drawing	Produce Engineering drawing
Manufacture	Define machine operation for manufacture
Cabling	Cable routing between connector & terminal
Cast	Design the assemblies & Prepare casting
Composite	Create the document part of composite material
Diagram	Create 2D Schematic representation
Die face	Design and analyze the contact surface
Format	Create and modify drawing format as per user
Lawet	Create 2D conceptual assembly sketches
Legacy	Import 3D data and 2D drawings
Mark Up	Make up a drawing, part or assembly
Mold	Create and analyze molds and modeling
Processors	St up CL data post processor
Process Report	Create custom report for assemblies
Scan Model	Create or dynamically modify surface
Sheet Metal	Create solid model of sheet metal parts
Verify	Compare scanned model and design data

Feature base modeling

The 'chunks' of solid material from which Solidworks mode are constructed, are called as features, Features generally fall into the following categories.

- Base feature



Base feature is a sketched feature or datum plane is important because all futures model geometry will reference this feature directly or indirectly.

- Sketch feature:

In general sketched feature are created by extruding, revolving, blending or sweeping a sketched cross-section.

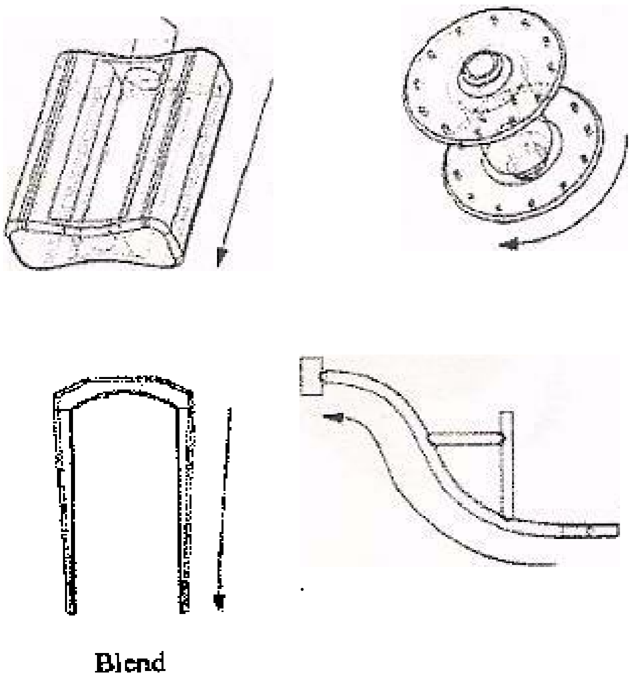


Figure 1:- Sketch Feature of solidsorks

- Reference feature:

Reference features references existing geometry and employ an inherent form, they do not be sketched. Reference feature are hole, shell, round, chamfered, draft etc

- Datum features:

Datum features such as planes, axis, curves and points are used to provide sketching planes and counter references for sketch and reference features. Datum features do not have physical volume or mass and may be visually hidden without affecting solid geometry.

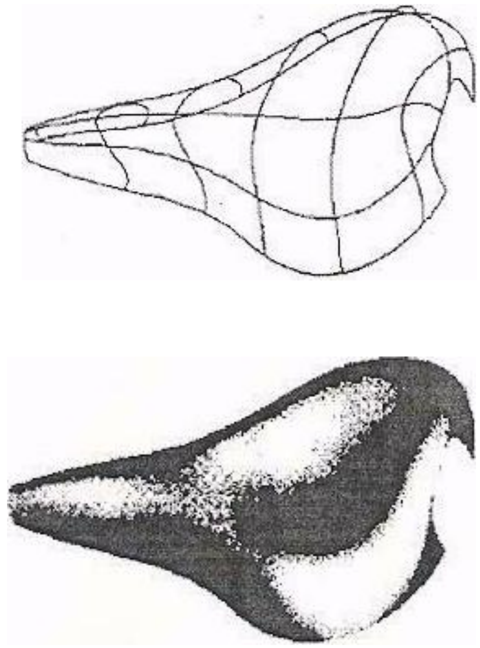


Figure 2:- Datum Features

- Parametric modifications

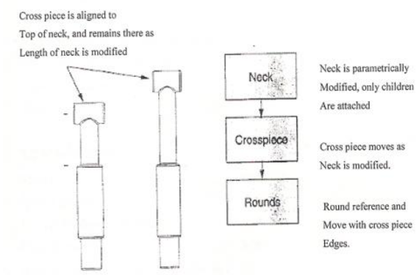
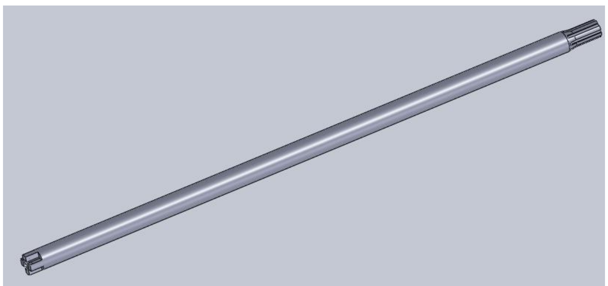
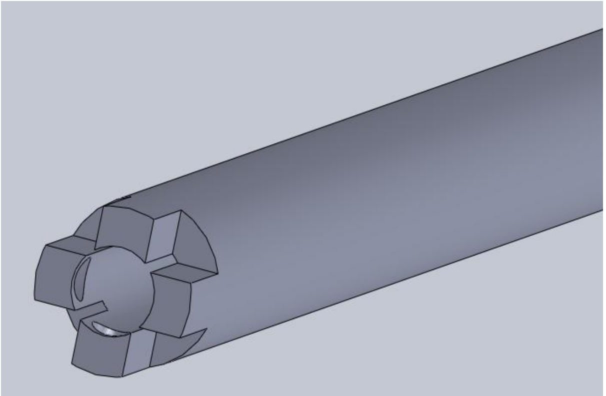


Figure3 :-Parametric Modifications

When a parent features are modified its child features are automatically modified. It is therefore essential to reference feature dimensions. So the design modifications are correctly propagated through a model.





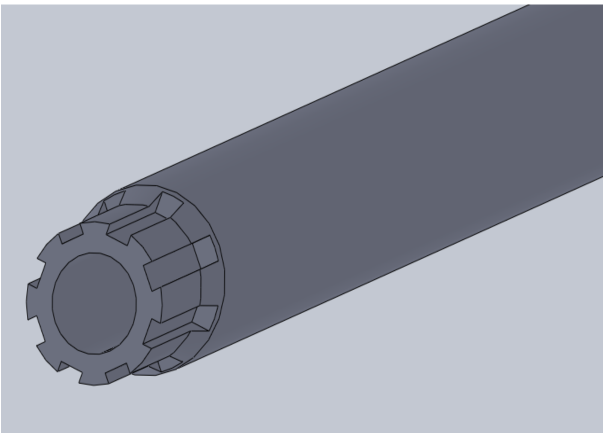
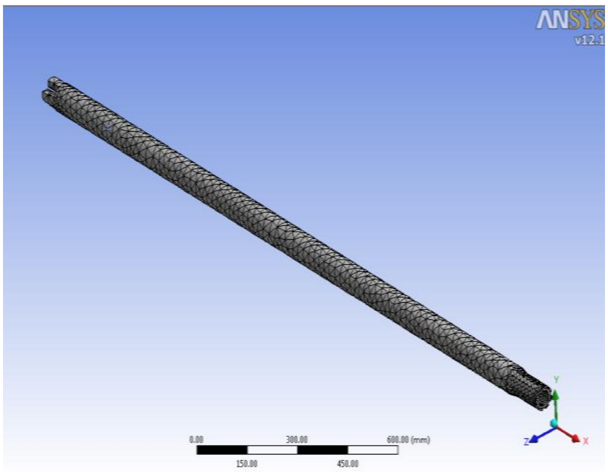


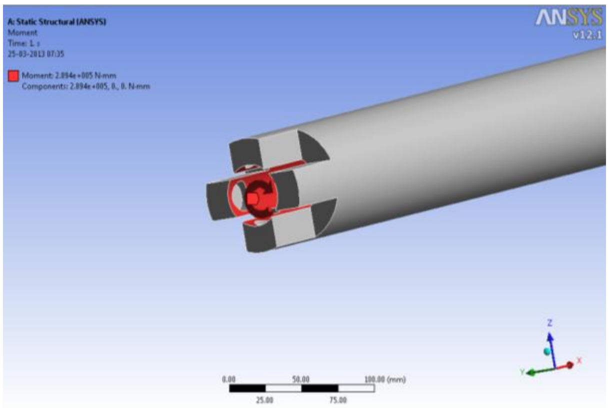
Figure 4:- FEM

- A numerical method
- Mathematical representation of actual problem
- Approximate method

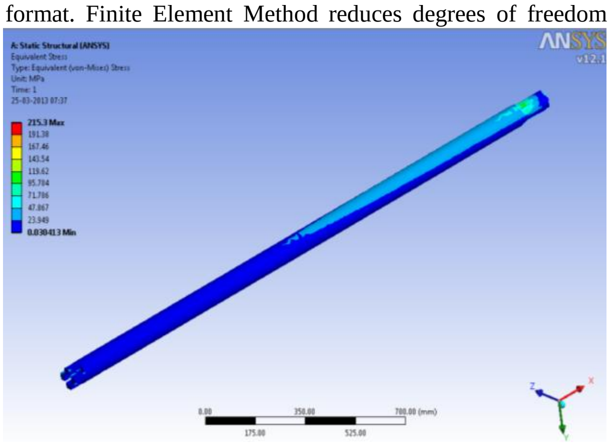
Definition of FEM is hidden in its words itself. Basic theme is to make calculations at only limited (Finite) number of points and then interpolate the results for entire domain (surface or volume).

Finite – Any continuous object has infinite degrees of





freedom & it's just not possible to solve the problem in this format. Finite Element Method reduces degrees of freedom



from Infinite to Finite with the help of discretization i.e. meshing (nodes & elements).

Element – All the calculations are made at limited number of points known as nodes. Entity joining nodes and forming a specific shape such as quadrilateral or triangular etc. is known as Element. To get value of variable (say displacement) any where in between the calculation points, interpolation function (as per the shape of element) is used.

Method –There are 3 methods to solve any engineering problem. Finite element analysis belongs to numerical method category.

Advantages of FEA

- ✓ Visualization increases.
- ✓ Design cycle time reduces
- ✓ No. of prototypes reduces
- ✓ Testing reduces
- ✓ Optimum design

3. ANALYSIS OF PROPELLER SHAFT USING ANSYS

In analysis first model is created then meshing is done and then loads are applied. The following pictures show the analysis of propeller shaft and corresponding stress induced.

Table 2:-Comparison of FE Analysis Results with Experimental Results:

	FE Analysis	Experimental	% Deviation
Von Misses Stresses(Mpa)	215.3	218.5	1.4645309

4. OPTIMIZATION

Different resins are used for optimizing the stress induced in propeller shaft made up of steel material. Analysis are as follows when different resins are used. Then comparison is given with the properties of resins.

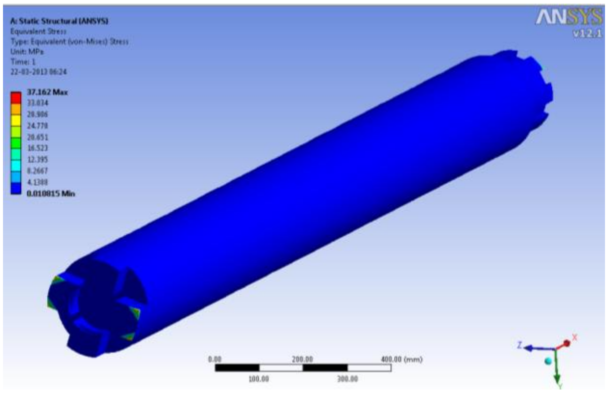


Figure 5:-Analysis of E-Glass Epoxy

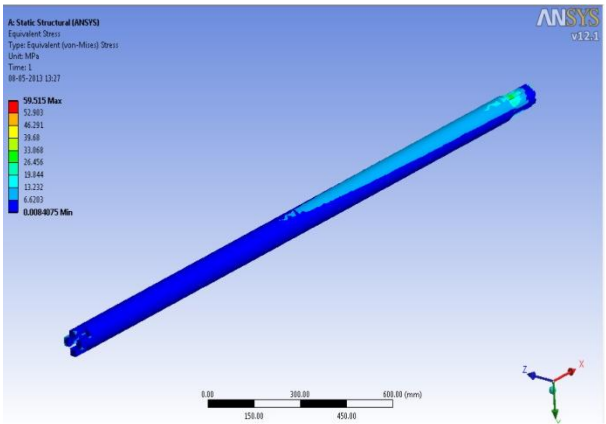


Figure 6:-Analysis of Kelvar Epoxy

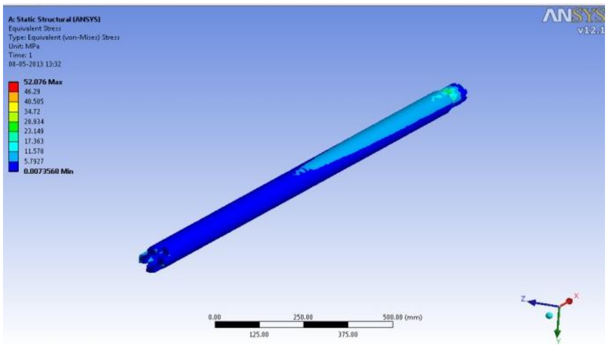
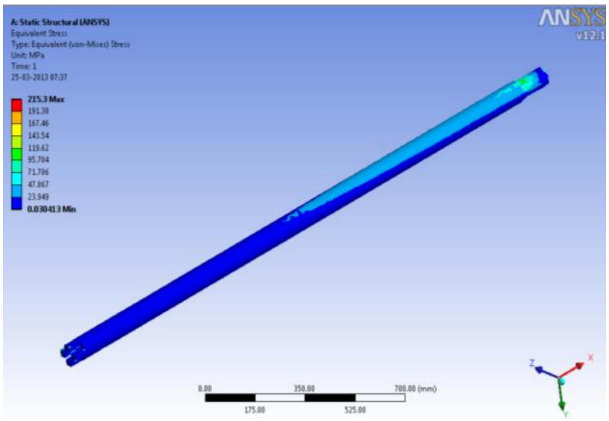


Figure 7:- Analysis of Boron Epoxy

Table 3. Comparison of Differential Material

Material	Von Misses Stresses(Mpa)
Steel	215.3
E-Glass/Epoxy	37.162
kelvar Epoxy	59.515
Boron Epoxy	52.076



Properties of Composite Material

EX, EY, and EZ is the modulus along X, Y, Z direction, respectively; υ_{xy}, υ_{yz}, and υ_{zx} is the Poisson's ratio along XY, YZ, and ZX direction, respectively; σ_t it the ultimate tensile strength; and σ_c is the ultimate compression strength.

Table 4. Composite Material Propety

Material	Ex (GPa)	Ey (GPa)	Ez (GPa)	υ_{xy}	υ_{yz}	υ_{zx}	σ_t (MPa)	σ_c (MPa)	ρ Density Kg/mm3
E-Glass/ epoxy	45.7	15.9	15.9	0.27	0.094	0.094	1203.67	703.25	2.6x10-6
Kelvar Epoxy	38.7	12.89	14.7	0.22	0.091	0.094	981.78	607.84	2.1x10-6
Boron Epoxy	41.4	13.87	15.1	0.25	0.092	0.089	1147.17	651.78	2.4x10-6

5. CONCLUSION

By using software ansys we can predict the mechanical properties of composite material like stresses without using less man power and expenses. There would be very less chance of material failure and any design fault.

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