International Journal of Research
Available at https://edupediapublications.org/journals
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue-17
December 2017

Modelling & Analysis of Propeller Blade

Dr. Rayapuri Ashok & Mr. Machani Ashok,
Associate Professor & Hod,M.Tech Scholor, Department Of Mechanical Engineering
Bheema Institute Of Technology & Science

ABSTRACT: *Fiber reinforced composites are finding wide spread use in naval applications in recent times. Ships and under water vehicles like torpedoes, Submarines etc., these weapons require propeller to drive the vehicle. In general the propeller will be used as propulsions and it also used to develop significant thrust to propel the vehicle at its operational speed. which are designed for moderate and deeper depths require minimization of structural weight for increasing payload, performance/speed and operating range for that purpose Aluminum alloy casting is used for the fabrication of propeller blades In current years the increased need for the light weight structural element with acoustic insulation, has led to use of fiber reinforced multi layer composite propeller. The present work carries out the structural analysis of a CFRP (carbon fiber reinforced plastic) propeller blade which proposed to replace the Aluminum propel blade. In addition propeller is subjected to an external hydrostatic pressure on either side of the blades depending on the operating depth and flow around the propeller also result in differential hydrodynamic pressure between face and back surfaces of blades. These hydrodynamic pressure takes as a single resultant load act at 1/3 part of the blade. The propeller blade is modeled and designed such that it can with stand the static and dynamic load distribution and finding the stresses and deflections in static analysis for both isotropic and orthotropic materials. In dynamic analysis free vibration (modal analysis) analysis for both isotropic and orthotropic materials. This thesis work basically deals with the modeling and design analysis of the propeller blade of a ship for its strength. A propeller is complex 3D model geometry. Which requires high end modeling CAD software is used for generating the blade model in CATIA V5 R19. This report consists of brief details about Fiber Reinforced Plastic materials and the advantages of using composite propeller over the conventional metallic propeller. This project concentrates on the metal and composite strength analysis of the propeller blade carried out by using the finite element method. By using ANSYS12.0 software static, modal were carried out for both isotropic and orthotropic material. In this work two different types of ship propeller i.e. Aluminum, Composite, were studied using FEM techniques. .*

Key Words: Strength of Propeller Blades, Materials, Modeling Of Propeller , Modal analysis .

I. INTRODUCTION

Propellers produce thrust through the production of lift by their rotating blades. Propeller hydrodynamics is therefore part of the broader field of lifting-surface theory, which includes such varied applications as aircraft, hydrofoil boats, ship rudders, and sailboat keels. Air and water propellers have much in common from a theoretical point of view, particularly if one's attention is restricted to air propellers operating at low Mach numbers (where compressibility effects are negligible) and to water propellers operating without cavitation.

The cross sections of most lifting surfaces are also similar in appearance, being designed to produce a force at right angles to their motion through the fluid (lift) with a minimum force parallel to their direction of motion (drag). In spite of these fundamental similarities, air and water propellers generally look very different. The reason is that propellers for ships are limited, for practical reasons, in diameter, and they are also limited by cavitation in the amount of lift per unit blade area that they can produce. As a result, marine propellers have blades that are much wider in relation to their diameter than would be found in aircraft propellers.

In addition, propellers are generally located in close proximity to the stern of a ship. This choice is based both on consideration of propulsive efficiency and on such practical matters as machinery arrangement and vulnerability to damage. Since the flow near the stern is nonuniform, an inevitable consequence is the development of vibratory forces on the propeller blades and on the hull. Decisions concerning the number of blades and the shape of the blade outline are influenced to a great extent by the need to minimize this excitation. As an example, a photograph of a recently designed propeller for a seismic exploration vessel.

1.1 Principle Of Propeller

- propeller are based on Bernoulli's principle¬ and Newton's third law
- Propeller works by throwing mass in the¬ opposite direction you want to go, which ,by new tons law produces equal and opposite reaction of you moving
- It based on push and pull concept¬

1.2. Strength of Propeller Blades

A first approach to strength problem was made by Taylor [1] who considered a propeller blade as a cantilever rigidly fixed at the boss. J E Connolly [2] addresses the problem of wide blades, tried to combine both theoretical and experimental investigations. The author carried out the measurements of deflection and stresses on model blades subjected to simulated loads with an aim to develop a theoretical model calibrated against the laboratory experiments. Terje sont vedt [3] studied the application of finite element methods for frequency response and improve to the frozen type of hydrodynamic loading.

1.3. Material selection

The stress-strain curve of the composite. The stress strain curve for a composite lies in between the stress strain curves of the fibers and matrix. The actual location of the composite stress strain curve will depend upon the relative volume fraction of the constituents. If the fibre volume fraction is high the composite stress strain curve will be close to the fibre stress-strain curve. On the other hand the composite stress–strain curve may be closer to the matrix stress-strain curve for a higher matrix volume fraction.

II. LITERATURE SURVEY

Benjamin viney et. Al. [1*]* has presented the flow around the enshrouded marine propellers operating in the wake of an axisymmetric body is rotational and tridimensional. An inverse method based on the model of inviscid and rotational fluid and coupling two complementary steps (axisymmetric computation +3D panel method) is proposed for the design of the marine propellers. The maridional flow computation leads to the determination of axisymmetric stream sheets as well as the approximate camber surface of the blades and gives a good estimation of the surface of the free vertex wake.. **J.E Connolly et al [2]:** Address the problem of wide blades tried to combine both theoretical and experimental investigations, the author carried out measurements of deflection and stress on models blades subjected to simulated loads, with an aim to develop a theoretical model calibrated against the laboratory experiments, the model was then validated by measurements of pressure and stress distribution on the blade of a full- scale ship propeller at sea. based on the experimental results it was concluded that wide blades subject to tensile stress strength on the face and compression stress of similar magnitude in the back was pointed out that the accuracy of the prediction from the modal depends on the accuracy of the working load determined.

Terge sont vcdt et al [3*]* has focused on the application of finite element methods for frequency response and improve to the frozen type of hydrodynamic loading The thin shell element of the triangular type and the super parametric shell element are used in the finite element model it presents the realistic an dynamic stresses in marine propeller blades. Stresses and deformations calculated for ordinary geometry and highly skewed propellers are compared with experimental results.

Chang suppler et al *[4]* have investigated the main sources of propeller blade failures and resolved problem systematically. An FEM analysis is carried out to determine the blade strength in model and full-scale condition and range of safety factor for the propeller under study is determined.

S.javed jalali and farid Taheri et al [5] Carbon fiber reinforced plastics properties were taken from journal of composite materials. A new test method for the simultaneous evaluation of the longitudinal and the shear module of CFRP was introduced under the proposed method, specimens with different span to depth ratios are subjected to three point bending method. Therefore, we name the method the varying span method. The method builds on the inherent low shear modulus characteristics of CFRP. This characteristics leads to a flexural modules which is a function of L/H.

Charles A. Harper et al [6] Aluminum material property taken from the hand book of material and process. The non-ferrous metals and alloys offer a wide variety of physical and mechanical properties for using the many industries. Aluminum and its alloys posse's properties which find wide use in the many industries. Favorable physical properties good strength- weight properties, good corrosion resistance, and low density. Combined with economy in material cost and fabrication cost, make this alloy family a basic material of construction for mechanical assemblies.

III. THEORY OF PROPELLER

Theory of propeller

The computational model used in its design, which we discuss later, is illustrated. The complex blade shape is required because this propeller must have very low levels of vibratory excitation and be completely free of cavitation under certain operating conditions. The complete field of marine propeller hydrodynamics is far too broad to cover adequately in a single paper. In this review we restrict our attention to single-unit propellors, as illustrated in. Multicomponent propellers consisting of pairs of counter rotating propellers, combinations of rotors and stators, or propellers combined with fixed or rotating shrouds are all of current interest but are not covered here. Propeller cavitation is an extensive field of its own, which we also do not cover except as a motivation for determining accurate pressure distributions on the blades.

However, the reader should be aware that computational techniques for noncavitating flows, which we do describe, have been extended to the case of cavitating flows. Recent work in this particular area is reviewed in Van Houten et al. (1983). Another important aspect of propeller hydrodynamics that we do not cover here is the interaction of the pressure field of the propeller with the hull. The published literature in this field is extensive, and the interested reader might possibly start with publications by Breslin et al. (1982), Vorus (1976), and ¥orus et al. (1978). In this review we first discuss the onset flow to the propeller, which must be known before one can proceed with the solution of the propeller problem. We then formulate briefly the problem of the flow around a propeller in general terms, at which point we look specifically at the problems of designing a propeller for a given distribution of lift, analyzing a given propeller both in circumferentially uniform flow and in the unsteady flow resulting from a nonuniform onset field.

Types of marine propellers

- Controllable pitch propeller
- Skewback propeller
- Modular propeller

Controllable pitch propeller

A controllable pitch propeller one type of marine propeller is the controllable pitch propeller. This propeller has several advantages with ships. These advantages include: the least drag depending on the speed used, the ability to move the sea vessel backwards, and the ability to use the "vane"-stance, which gives the least water resistance when not using the propeller (e.g. when the sails are used instead).

Skewback propeller

An advanced type of propeller used on German Type 212 submarines is called a skewback propeller. As in the scimitar blades used on some aircraft, the blade tips of a skewback propeller are swept back against the direction of rotation. In addition, the blades are tilted rearward along the longitudinal axis, giving the propeller an overall cup-shaped appearance. This design preserves thrust efficiency while reducing cavitation's, and thus makes for a quiet, stealthy design. I

Modular propeller

A modular propeller provides more control over the boats performance. There is no need to change an entire prop, when there is an opportunity to only change the pitch or the damaged blades. Being able to adjust pitch will allow for boaters to have better performance while in different altitudes, water sports, and/or cruising.

3.1Three Bladed Right Hand Propeller

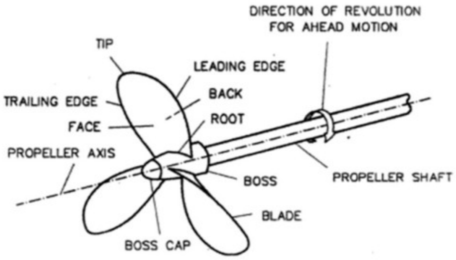


Fig 1 : Three Bladed Right Hand Propeller

3.2 Design Parameters

we consider a propeller consisting of K identical, symmetrically arranged blades attached to a hub that is rotating at constant angular velocity ~o about the x-axis. The hub is either idealized as an axisymmetric body as shown or ignored completely.

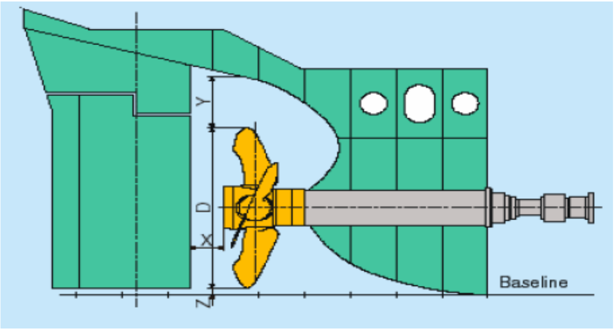


Fig.2: Design Parameters

The geometry of the blades and hub is prescribed in a Cartesian coordinate system rotating with the propeller. The y-axis is chosen to pass through the midchord of the root section of one blade, which we designate the key blade. Tale z-axis completes the right-handed system. An equivalent cylindrical coordinate system in which r is the radial coordinate and 0 = 0 on the y-axis is also used here. www.annualreviews.org/aronline Annual Reviews Annu. Rev. Fluid Mech.

MARINE PROPELLERS 375 The blade is formed starting with a midchord line defined parametrically by the radial distribution of skew angle 0m(r) and rake Xm(r). By advancing distance +½e(r) along a helix of pitch angle cpp(r), one obtains the blade leading edge and trailing edge, respectively, and the surface formed by the helical lines at each radius form the reference upon which the actual blade sections can be built. These sections can be defined in standard airfoil terms by a chordwise distribution of camber f(s) and thickness t(s), where s is curvilinear coordinate along the helix.

3.3 Propeller Blade Cylindrical Section

The propeller is operating in an unbounded, incompressible fluid, in a prescribed effective onset flow, as described in the preceding section. This flow is defined in a fixed coordinate system in which the x and Xo axes are identical, and the y and Yo axes are coincident at time t = 0. If we ignore the variation of the effective onset flow, both with respect to time and longitudinal position Xo, and make use of the cyclic nature of the flow, we can write down the velocity components in the following generally accepted

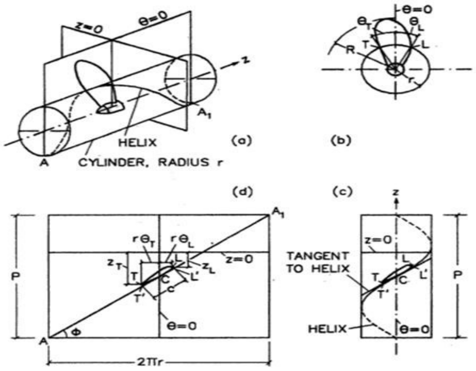


Fig.3 propeller blade cylindrical section

3.4 Effect Of Pitch

Pitch converts the torque of the propeller shaft to thrust by deflecting or accelerating water as tern. The fundamental task in selecting a propeller is tochoo sea pitch and diameter that will generate the maximum thrust possible at normal operating .

Speed without overloading the engine Increasing pitch increases thrust but increasing pitch too much reduces the efficiency of the engine and propeller combination by slowing the engine.

On the other hand, while too little pitch will not over loaders low the engine, it will not accelerate as much water as tern and thus will not generate maximum possible thruster speed.

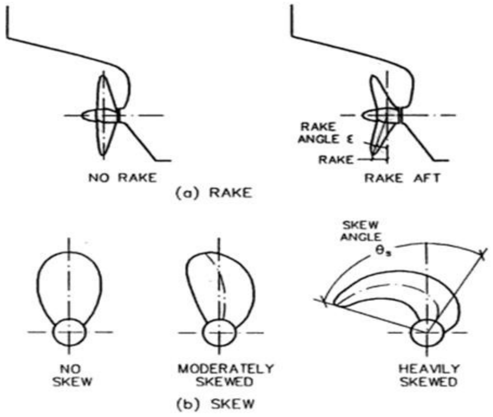


Fig .4. EFFECT OF PITCH

IV. CATIA

CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Assault Systems. Written in the C++ programming language, CATIA is the cornerstone of the Assault Systems product lifecycle management software suite.

Developer(s)	Dassault Systems
Stable release	V6R2011x / November 23, 2010
Operating system	Unix / Windows
Type	CAD software
License	Proprietary
Website	WWW.3ds.com

Table 4.1: Details of CATIA

4.1 History Of Catia

CATIA started as an in-house development in 1977 by French aircraft manufacturer Avions Marcel Dassault, at that time customer of the CADAM CAD software to develop Dassault's Mirage fighter jet, then was adopted in the aerospace, automotive, shipbuilding, and other industries.

Initially named CATI (Conception Assisted Tridimensionnelle Interactive - French for Interactive Aided Three-dimensional Design) - it was renamed CATIA in 1981, when Dassault created a subsidiary to develop and sell the software, and signed a non-exclusive distribution agreement with IBM.

In 1984, the Boeing Company chose CATIA as its main 3D CAD tool, becoming its largest customer.

In 1988, CATIA version 3 was ported from mainframe computers to UNIX.

In 1990, General Dynamics Electric Boat Corp chose CATIA as its main 3D CAD tool, to design the U.S. Navy's Virginia class submarine.

In 1992, CADAM was purchased from IBM and the next year CATIA CADAM V4 was published. In 1996, it was ported from one to four UNIX operating systems, including IBM AIX, Silicon Graphics IRIX, Sun Microsystems SunOS and Hewlett-Packard HP-UX.

In 1998, an entirely rewritten version of CATIA, CATIA V5 was released, with support for UNIX, Windows NT and Windows XP since 2001.

In 2008, Dassault announced and released CATIA V6. While the server can run on Microsoft Windows, Linux or AIX, client support for any operating system other than Microsoft Windows is dropped.

4.1.1 Release History

Name/Version	Latest Build Number	Original Release Date	Latest Release Date
CATIA v4	R25	1993	January 2007
CATIA v5	R20	1998	February 2010
CATIA v6	R2012	29/05/2008	May 2011

Table 4.2: versions of CATIA

4.2 Scope Of Application

Commonly referred to as 3D Product Lifecycle Management software suite, CATIA supports multiple stages of product development (CAx), from conceptualization, design (CAD), manufacturing (CAM), and engineering (CAE). CATIA facilitates collaborative engineering across disciplines, including surfacing & shape design, mechanical engineering, equipment and systems engineering

4.2.1 Surfacing & Shape Design

CATIA provides a suite of surfacing, reverse engineering, and visualization solutions to create, modify, and validate complex innovative shapes. From subdivision, styling, and Class A surfaces to mechanical functional surfaces.

4.2.2 Mechanical Engineering

CATIA enables the creation of 3D parts, from 3D sketches, sheet metal, composites, and molded, forged or tooling parts up to the definition of mechanical assemblies. It provides tools to complete product definition, including functional tolerances, as well as kinematics definition.

4.2.3 Equipment Design:

CATIA facilitates the design of electronic, electrical as well as distributed systems such as fluid and HVAC systems, all the way to the production of documentation for manufacturing.

4.2.4 Systems Engineering

CATIA offers a solution to model complex and intelligent products through the systems engineering approach. It covers the requirements definition, the systems architecture, the behavior modeling and the virtual product or embedded software generation. CATIA can be customized via application programming interfaces (API). CATIA V5 & V6 can be adapted using Visual Basic and C++ programming languages via CAA (Component Application Architecture); a component object model (COM)-like interface.

Although later versions of CATIA V4 implemented NURBS, V4 principally used piecewise polynomial surface. CATIA V4 uses a non-manifold solid engine.

CATIA V5 features a parametric solid/surface-based package which uses NURBS as the core surface representation and has several workbenches that provide KBE support.

V5 can work with other applications, including Enova, Smarteam, and various CAE Analysis applications.

4.3 Supported Operating Systems And Platforms

CATIA V6 runs only on Microsoft Windows and Mac OS with limited products.

CATIA V5 runs on Microsoft Windows (both 32-bit and 64-bit), and as of Release 18Service Pack4 on Windows Vista 64.IBM AIX, Hewlett Packard HP-UX and Sun Microsystems Solaris are supported.

CATIA V4 is supported for those Unixes and IBM MVS and VM/CMS mainframe platforms up to release 1.7.

CATIA V3 and earlier run on the mainframe platforms.

4.4.Ship building

Dassault Systems has begun serving shipbuilders with CATIA V5 release 8, which includes special features useful to shipbuilders. GD Electric Boat used CATIA to design the latest fast attack submarine class for the United States Navy, the Virginia class. Northrop Grumman Newport News also used CATIA to design the GeraldR. Ford class of super carriers for the US Navy.

4.4.1. Industrial Equipment

CATIA has a strong presence in the Industrial Equipment industry. Industrial Manufacturing machinery companies like Schuler and Metro use CATIA , as well as heavy mobile machinery and equipment companies like Class, and also various industrial equipment product companies like Alstom Power and ABB Group.

4.4.2. Other

Architect Frank Gehry has used the software, through the C-Cubed Virtual Architecture company, now Virtual Build Team, to design his award-winning curvilinear buildings. His technology arm, Gehry Technologies, has been developing software based on CATIA V5 named Digital Project. Digital Project has been used to design buildings and has successfully completed a handful of projects.

STEP 1: OPEN CATIA V5

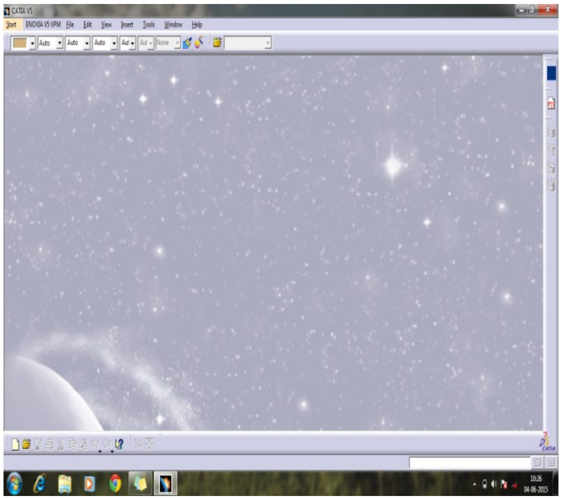


Fig 4.1: OPEN CATIA V5

STEP 2: Open Start→shape →Generative Shape Design

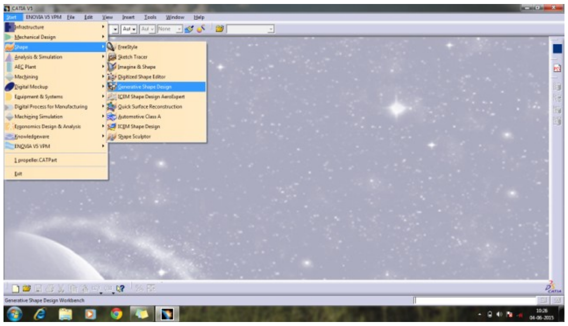


Fig 4.2: Generative Shape Design

STEP 3: Generative Step design:

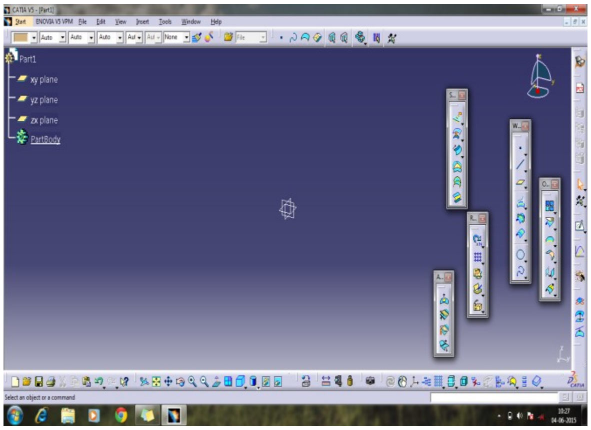


Fig 4.3: Generative Step design

STEP 4 : Tools

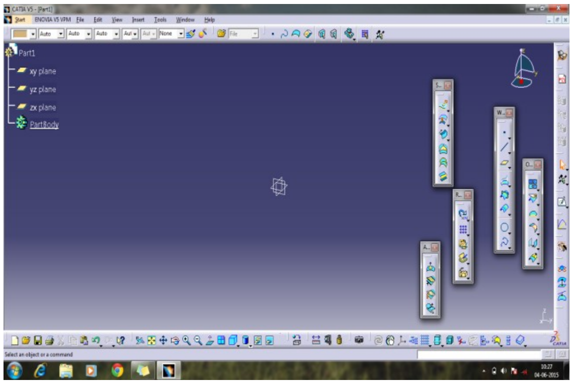


Fig 4.4: Tools

STEP 5: Draw a circle with 15 radius.

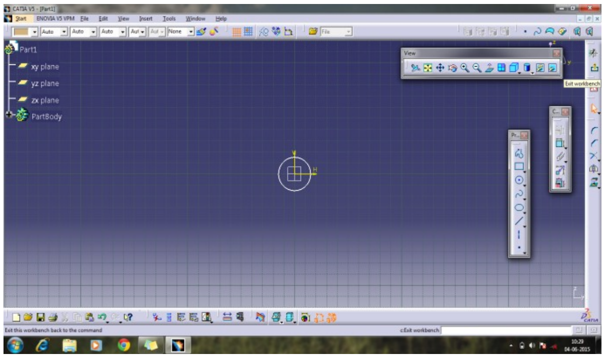
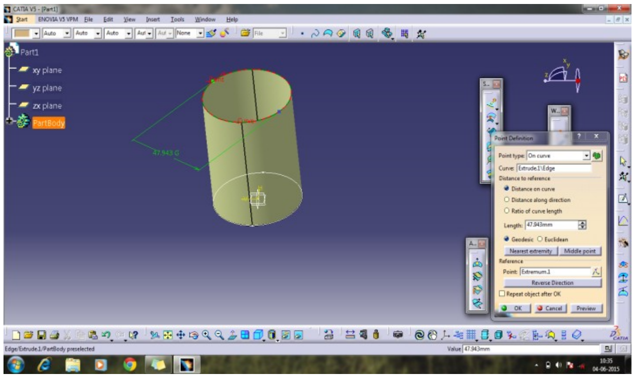


Fig 4.5: Draw a circle with 15 radius.

STEP 6 :Using Extrude Command Extrude 40 mm by using point command select a point on a curve



Point type: On curve
Curve: Extrude.1\Edge
Distance to reference
● Distance on curve
○ Distance along direction
○ Ratio of curve length
Length: 47.943mm
● Geodesic ○ Euclidean
Reference
Point: Extremum.1
☐ Repeat object after OK

Fig 4.6: Using Extrude Command Extrude 40 mm by using point command select a point on a curve

V. ANSYS

5.1 History:

The company was founded in 1970 by Dr. John A Swanson Analysis systems, Inc. SASI. Its primary purpose was to develop and market finite element analysis software for structural physics that could simulate static (stationary), dynamic (moving) and heat transfer (the rmal) problems. SASI developed its business in parallel with the growth in computer technology and engineering needs. The company grew by 10 to 20 percent year, and in 1994 it was sold to TA Associates. The new owners took SASI's leading software called ANSYS as their flagship product and designated ANSYS, Inc as the new company name.

5.2 Introduction

The ANSYS program is a computer program for a finite element analysis and design. The ANSYS program can also be used to calculate the optimal design for given operating conditions using the design optimization feature.

ANSYS is commercial finite-element analysis software with the capability to analyze a wide range of different problems. ANSYS runs under a variety of environments, including IRIX, Solaris, and Windows NT. Like any finite-element software, ANSYS solves governing differential equations by breaking the problem into small elements. The governing equations of elasticity, fluid flow, heat transfer, and electro-magnetism can all be solved by the Finite element method in ANSYS. ANSYS can solve transient problems as well as nonlinear problems. This document will focus on the basics of ANSYS using primarily structural examples.

The ANSYS program is a multi-purpose program, meaning that you can use it for almost any type of finite element analysis in virtually any industry - automobiles, aerospace, railways, machinery, electronics, sporting goods, power generation, power transmission, and biomechanics, to mention just a few. “Multi-purpose” also refers to the fact that the program can be used in all disciplines ofengineering – structural, mechanical, electrical, electromagnetic, electronic, thermal, fluid, and biomedical. The ANSYS program is also used as an educational tool in universities and other academic institutions.

ANSYS is available on all MEnet Sun and SGI machines. It is available on the Linux machines by remote-login only. On the right side, rumor has it that ANSYS is looking into a Linux port. Currently, MEnet uses the Research/Faculty version of ANSYS 12.1. The Research/Faculty license level permits larger, more complex models than does the current level running on the IT Labs machines. This document is meant to be a starting point. The material covered here is by no means comprehensive. In fact, we will only scratch the surface of ANSYS's capabilities. Given that, I will try to cover most of what I know about ANSYS and some tricks I have learned while using it. The document will begin with two simple examples, taking the user through all of the steps of creating a model, meshing, adding boundary conditions, solving, and, finally, looking at the results. The remainders of this document will over tips and tricks for each of the steps.

5.3 Basic Methodology Of Ansys

ANSYS is followed up by the method called Finite Element Modelling Methods(FEM)\

5.3.1 Finite element method

The finite element method (FEM) (its practical application often known as finite element analysis (FEA) is a numerical technique for finding approximate solutions of partial differential equations (PDE) as well as of integral equations. The solution approach is based either on eliminating the differential equation completely (steady state problems), or rendering the PDE into an approximating system of ordinary differential equations, which are then numerically integrated using standard techniques such as Euler's method, Runge-Kutta, etc.

In solving partial differential equations, the primary challenge is to create an equation that approximates the equation to be studied, but is numerically stable, meaning that errors in the input and intermediate calculations do not accumulate and cause the resulting output to be meaningless. There are many ways of doing this, all with advantages and disadvantages. The Finite Element Method is a good choice for solving partial differential equations over complicated domains (like cars and oil pipelines), when the domain changes (as during a solid state reaction with a moving boundary), when the desired precision varies over the entire domain, or when the solution lacks smoothness. For instance, in a frontal crash simulation it is possible to increase prediction accuracy in "important" areas like the front of the car and reduce it in its rear (thus reducing cost of the simulation). Another example would be in Numerical weather prediction, where it is more important to have accurate predictions over developing highly-nonlinear phenomena (such as tropical cyclones in the atmosphere, or eddies in the ocean) rather than relatively calm areas.

5.3.3 Mouse

Left mouse button picks (or unpicks) the entity or location closest to the mouse pointer. Pressing and dragging allows you to “preview” the item being picked (or unpicked).Middle mouse button does an Apply. Saves the time required to move the mouse over to the Picker and press the Apply button. Use Shift-Right button on a two-button mouse.

Right mouse button toggles between pick and unpick mode. Note, the Shift-Right button on a two button mouse is equivalent to the Middle mouse button on a three-button mouse.

5.3.4 Database And Files

The term ANSYS database refers to the data ANSYS maintains in memory as you build, solve, and post process your model. The database stores both your input data and ANSYS results data:

–Input data -- information you must enter, such as dimensions, material properties, and load data.

–Results data -- quantities that ANSYS calculates, such as displacements, stresses and temperature.

5.3.5 File Management Tips

- Run each analysis project in a separate working directory.
- Use different job names to differentiate various analysis runs.
- You should keep the following files after any ANSYS analysis:

log file (.log); database file (.db); results files (.rst…); load step files, if any (.s01, .s02, ...)

Defining an Analysis Title: Utility Menu> File> Change Title

This will define a title for the analysis. ANSYS includes the title on all graphics displays and on the solution output. (Please include your name and student ID in the analysis title for all original graphs)

5.4 Ansys Graphical User Interface (Out Put Window)

After starting ANSYS, two windows will appear. The first is the ANSYS 8.1 Output Window:

This window displays a listing of every command that ANSYS executes. If you encounter problems, this is a good place to look to see what ANSYS is doing or has done. This is one location where you will find all of the warnings and error messages that appear and the command that generated the warning/error.

The second window is the ANSYS Research FS graphical user interface. This is divided into 4 sections (shown on next page):

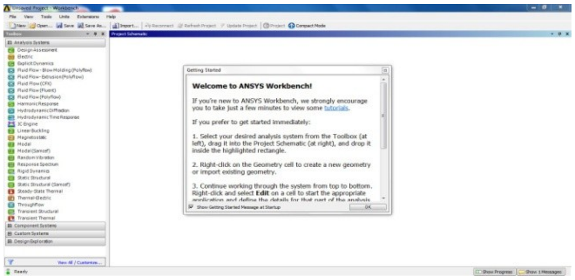
- ANSYS Utility Menu
- ANSYS Toolbar Menu
- ANSYS Main Menu
- Display window

5.5 Begin To Work In Ansys:

In the Graphic User Interface of ANSYS 14.5, you can find ANSYS main menu, there go for Preferences. A dialogue box preference for GUI filtering appears, select STRUCTURAL and click ok.

5.5.1 Begin the work in ANSYS WORKBENCH

Start the ANSSY 14.5 WORKBENCH from the desktop by double click on it. The start-up screen will look like this.



Welcome to ANSYS Workbench!

If you're new to ANSYS Workbench, we strongly encourage you to take just a few minutes to view some tutorials.

If you prefer to get started immediately:

1. Select your desired analysis system from the Toolbox (at left), drag it into the Project Schematic (at right), and drop it inside the highlighted rectangle.

2. Right-click on the Geometry cell to create a new geometry or import existing geometry.

3. Continue working through the system from top to bottom. Right-click and select **Edit** on a cell to start the appropriate

☑ Show Getting Started Message at Startup

OK

Fig .5.1: Start-Up Screen of Ansys Workbench

After selecting Structural define type of element in Preprocessor, select Add/edit, a dialogue box namely Element type appears, click on button ADD and select Solid 10 node 187 in library of element type and then click ok.

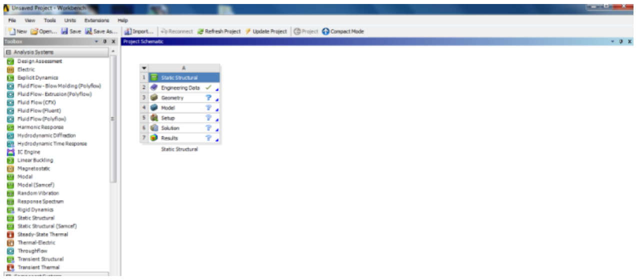


Fig.5.2: After selecting Structural define type of element in Preprocessor

- Importing blade from catia v5 r20

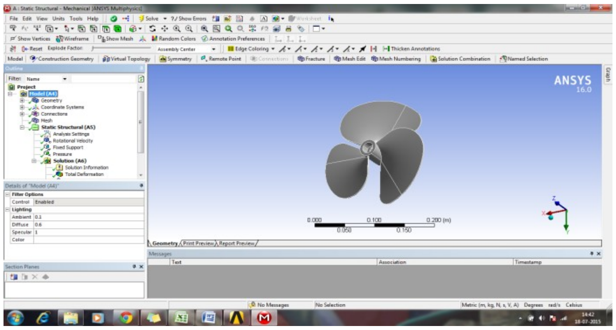


Fig 5.3: Importing blade from catia v5 r20

- Fine meshing view

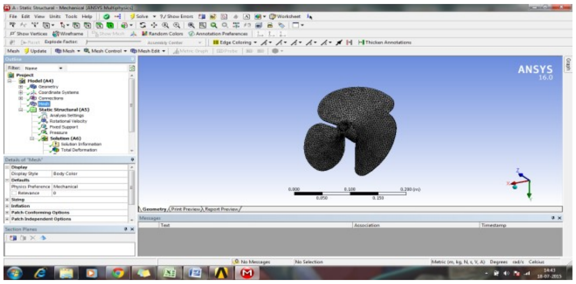


Fig.5.4: Fine meshing view

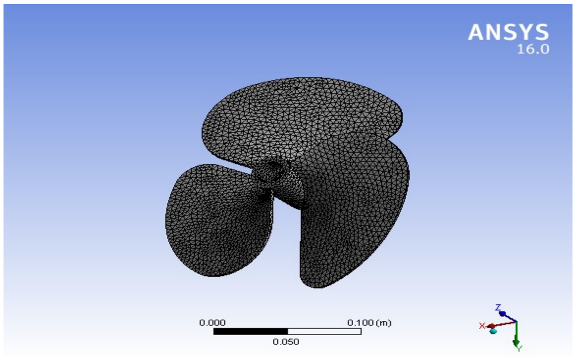


Fig.5.5: Fine meshing final view

- Applying rotation velocity factors

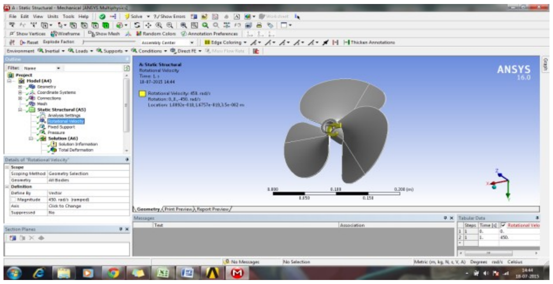


Fig.5.6: Applying rotation velocity factor

VI. CONCLUSION

Propeller creates a high pressure and low pressure volume which helps vessels to move in forward direction.

Design of propeller blade plays an important role. So we carefully designed propeller blades with required dimensions using CAD TOOL SOFTWARE namely CATIA V5R20 which has many advanced tools which helps for accurate design

Wireframe and surface design workbench is employed from CATIA V5 R20 which has many single axis tools which is used for design.

Excusing material tools are employed from part design workbench.

STRUCTURAL ANALYSIS is done in ANSYS 16 WORKBENCH under required boundary condition and followed by CFD ANALYSIS is done in fluent workbench.

We get maximum 1.108e3 velocity which is a pretty good velocity and we get pressure difference of 3.561e8 by which there is a large pressure difference when compared to its minimum pressure values..

By basic pressure difference the whole body moves in the region where the pressure is low so in this criteria body moves.

REFERENCES

[1] Bloor, M. I. G. and Wilson, M.J. Generating blend surfaces using partial differential equations. CAD, 21(3):165-171, 1989.

[2] Mortenson, M. E. Geometric modelling. Wiley-Interscience, New York, 1985.

[3] Kerwin, J. E. and Lee, C-S. Prediction of steady and unsteady marine propeller performance by numerical lifting-surface theory. Trans. Society of Naval Architects and Marine Engineers, 86, 1978.

[4] Hess, J. L. Calculation of potential flow about arbitrary three-dimensional lifting bodies. Technical Report MDCJ5679-01, McDonell Douglas, Oct 1972.

[5] Friesch, J. Possibilities of model tests for energy saving devices. In Marine Jubilee Meeting, Wageningen, The Netherlands, 1992.

[6] Young, F. R. Cavitation. McGraw-Hill Book Company Limited, Maidenhead, England, 1989.

[7] Rossignac, A. R. and Requicha, A. A. G. Constant radius blending and solid modelling. Computers in Mechanical Engineering, pages 65-73, 1984.

[8] Elliot, W. S. Computer-aided mechanical engineering: 1958 to 1988. CAD,21(5):274- 288,1989.

[9] Bezier, P. Style, Mathematics and NC. CAD, 22(9):524-526, 1990.

[10] Woodwark, J. Computing shape. Butterworths, London, 1986.