

Stress Analysis on a Turbine Shaft

¹Dr. M. Ramakotaiah ²M Sudheer Kumar ³T Ramesh Babu ⁴T Ravi Teja

¹ Assoc. Prof., department of mechanical, sai tirumala NVR engineering college, Narasaraopeta.

² Asst. Prof., department of mechanical, sai tirumala NVR engineering college, Narasaraopeta

³ Asst. Prof., department of mechanical, sai tirumala NVR engineering college, Narasaraopeta

⁴Assoc. Prof., department of mechanical, sai tirumala NVR engineering college, Narasaraopeta.

Abstract— In engineering practice, the machine parts are subjected to various forces, which may be due to:

1. Energy transmitted
2. Inertia of reciprocating parts
3. Lack of balance of moving parts

The different forces acting on a machine parts produce various types of stresses.

The shaft is a component, which transmits rotation from one component to other. While transmitting it is subjected to many forces and stresses.

The stiffness of constructions is determined by modulus of elasticity of the material, geometrical characteristics of cross sections and linear dimensions of supports.

This project is an attempt to analyze the stresses developed to see whether the designed component is within the safe limits. The analysis is performed conventionally and checked with an analysis software ANSYS 8.0

Keywords-FEM, CAD, ANSYS, CATIA, FEA, Mechanics of solids,

1. FINITE ELEMENT METHOD

The finite element method has become a powerful tool for the numerical solution in engineering problems. Applications range from deformation and stress analysis of automotive, transmission systems, aircraft building and bridge structures and field analysis of heat flux, fluid flow, magnetic flow seepage and other flow problems. The basic idea in the finite element method is to find the solution of complicated problem by replacing it by a simpler one. In this method of analysis, a complex rein defining a continuum is discretize in to simple geometric shapes called finite elements. The material properties and the governing relationships are considered over these elements and express, duly considering the loading and constraints, results in assort of equations. Solution of these equations gives us the approximate behavior of the continuum. With the advantages in the computer technology and CAD system, complex model can be solved with relative ease. Several alternative configurations can be tried out on a computer before the first prototype is built. All of this suggests that we need to keep pace with these developments by understanding basic theory, modelling techniques and computational aspects of the finite element method. First of all it is necessary to discretize the solution region that is to convert it in to a system with finite number of unknowns so that the problem can be solved numerically. The discretization of the solution region has dome in different ways Secondly, the solution region is divided in to finite elements by imaginary lines, the connection to the different elements being made at the nodes only. This method is suitable for bodies with non-homogeneous material properties or with complex irregular

boundaries. The elements are assumed to be interconnected at a distinct point called nodes suited on the element boundaries. Finally a displacement in terms of nodal displacement parameters or using generalized coordinates is chosen to represent the displacement pattern of field with each element. Based on a function element stiffness matrix is generated and the stresses are computed.

FUTURE SCOPE:

The Crane hook can also be manufactured by using different composite materials with the developments In the field of material science and metallurgy. Also the load earning capacity of the crane hook can be greatly improved by making suitable modifications in the design using computer aided design and then performing analysis on the design to make sure whether the design is safe and also to establish the various loading conditions and determine the maximum load carrying capacity of the crane hook. A series of improvements in the design can be done by determining the most suitable cross section of the crane hook under the given loading conditions by performing design optimization Also the improvements can be made in the manufacturing process by selecting the best one which suites the given specifications of the design. This reduces the manufacturing lead time, improves the efficiency of product along with the reduction in the overall cost of the product which further leads to greater profits for the organization.

2. GENERAL PROCEDURE OF FINITE ELEMENT METHOD:

The solution continuum problem by finite element method usually follows a step-by-step process. The following steps shown in general how the finite element works.

1. Discretize the given continuum:

The essence of finite element method is to divide a continuum that is problem domain in to quasi-disjoint non-over lapping elements. This is achieved by replacing the continuum by a set of key points called nodes, which when connected properly produce the elements. The connection of nodes and elements form the finite element mesh. There are many varieties of element shapes and types are available. The number of nodes and elements that can be used in the problem is the matter of engineering judgment. As a rule, the number of nodes and elements, the more accurate the finite element solution, but also more expensive and more tome in needed to obtain a solution.

2. Select the domain approximation:

The variation of field variable in the problem is approximated within each element by a polynomial. The field variable may be scalar or vector. Polynomial is usually used to

approximate the solution over an element domain because they are easy to integrate and differentiate. The degree of polynomial depends on number of nodes per elements, the number of unknowns (field variables) at each node, and certain continuity requirements along the boundaries.

3. Develop element matrices and equations:

Once the nodes material problems of a given element are defined its corresponding matrices and equations can be defined. Four methods available to derive the element matrices and equations: the direct method, the variation method, the weighted residual method and the energy method.

4. Solve the unknowns at nodes:

The global system of algebraic equations solved via gauss eliminations method to provide the value of field variable at the nodes of finite element mesh. Values of field variables and their derivatives at the nodes form the complete finite element solution of the original continuum problem before discretization. Values at other points inside the continuum other than nodes are possible to obtain although it is not customarily done.

5. Interpret the results:

The final step is to analyze the solution and results obtained from previous step to make design decisions. The correct interpretation of these results require a sound background in both engineering and FEA. That is why treating FEA and FEM codes as a black box is usually not recommended and in fact it is considered as dangerous.

WORKING WITH ANSYS 8.0

One of the reasons for mode applications of the finite element method is due to the availability of number of package programs. Ansys package is one of the FEA software. Ansys program is computer program for the finite element analysis and design the program is used to find out how a design works under operating conditions. It also used to calculate the proper design for given operating conditions. From the computational point of view the five basic steps in the finite element analysis, can be well organized in three basic components.

- (a) Pre-processor: Discretization.
- (b) Processor: Computation of element properties. Assemblage of elements. Solution of equations of equilibrium.
- (c) Post-processor: Computation of stresses.

The requirements of pre and post-processor are described in the following section. Pre – processor and post – processor:

Finite element analysis of practical problems requires large amount of input data. Manual preparation of input data is a tedious, time consuming and error-prone task particularly for three –dimensional analysis of solids. In computer aided design, good and acceptable design is arrived at only after a few cycle of analysis and design procedure. Hence it will be advantageous to have software, which will aid in the preparation of input data and also in the interpretation of analysis results. Such programs are called pre and post-processors to FEA package. A pre-processor creates the finite element model and the input necessary for a finite element analysis. A post – processor accepts the result of analyses and generates tables, diagrams/pictures etc., for proper interpolation of results. General requirements of pre -processors and

post-processors: The pre-processors take minimum input from the user, create the finite element mesh and other data required for the analysis and display the model for data-check and correction, if any to be made by the user in interactive mode Graphical post processing of results helps to perceive the physical consequences of the analysis. Both the programs should be user oriented graphics hence easy to use with minimum instructions.

A large portion of the input data to the finite element analysis is concerned with geometric idealization of the structure and mesh generation. Automatic generation of finite element meshes for two and three-dimensional structures not only reduce a good percentage of manpower expenditure of the total analysis time but minimizing the error. Graphics oriented edit facility should be available to edit and display the finite element model and also enable the user to input additional data like loads boundary conditions etc.

The model values of the field variables are the intermediate results of finite element analysis, in such cases, the post processor has to work out the higher order derivatives of the field variables with reliable accuracy which is of interest in mechanical problems. The field variables usually displacements and stress/strain results which are of interest in many design problems. The post processor helps by means of various graphical outputs, to identify the critical values and regions for designing, and to decide up on the adequacy of modeling/designing and need for remodeling/redesigning the component/structure and solid.

TYPES OF ANALYSIS:

There are different types of analysis that can be performed using finite element analysis software. Some of them are:

1. Structural analysis.
2. Coupled-field analysis.
3. Electromagnetic analysis.
4. Fluids/ computational fluid dynamic analysis.
5. Thermal analysis.

STRUCTURAL ANALYSIS

Definition of structural analysis

Structural analysis is probably the most common application of the finite element method. The term structural(or structure) implies not only civil engineering structures such as bridges and buildings, but also naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts, and tools.

Types of structural analysis

The seven types of structural analyses available in the ANSYS family of products are explained below, the primary unknowns (nodal degrees of freedom) calculated in structural analysis are displacements.

Other quantities, such as strains, stresses, and reaction forces, are then derived from the nodal displacements.

Structural analysis available in the ANSYS Multiphysics, ANSYS mechanical, ANSYS Structural, and ANSYS Professional programs only.

You can perform the following types of structural analyses. Each of these analyses is discussed in detail.

Static Analysis—Used to determine displacements, stresses, etc. Under static loading conditions, both linear and nonlinear static analysis. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

Model Analysis--- Used to calculate the natural frequencies and mode shapes of a structure. Different mode extraction methods are available,

Harmonic Analysis--- Used to determine the response of structure harmonically time-varying loads.

Transient dynamic analysis— used to determine the response of a structure to arbitrarily time-varying loads. All non-linearity mentioned under static Analysis above are allowed.

Spectrum Analysis-- An extension of the modal analysis, used to calculate stresses and strains due to a response spectrum or a PSD input (random vibrations).

Buckling Analysis— used to calculate the bulking loads and determine the bulking mode shape. Both linear (Eigen value) bulking and nonlinear buckling analyses are possible.

Explicit Dynamic Analysis – This type of structural analysis is only available in the ANSYSLS- DYNA program. ANSYSLS-DYNA provides an interface to the LS-DYNA explicit finite element program. Explicit dynamic analysis is used to calculate fast solutions for large deformation dynamics and complex contact problems.

In addition to the above analysis types, several special-purpose features are available:

- . Fracture mechanics
- . Composites
- . Fatigue
- . P-Method
- . Beam analysis

SOLIDS95 Element Description:

SOLIDS95 is a higher order version of the 3-D 8-node solid element SOLID45. It can tolerate irregular shapes without as much loss of accuracy. SOLID95 elements have compatible displacement shapes and are well suited to model curved boundaries.

The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y and z directions. The element may have any spatial orientation. SOLID95 has plasticity, creep, stress stiffening, large strain capabilities. Various printout options are also available. SOLID95 in the ANSYS Inc Theory Reference for more details.

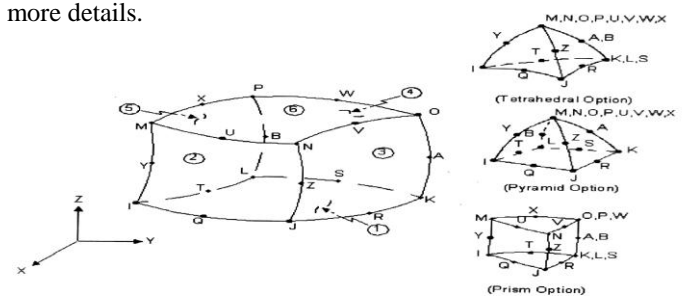


Figure 1. SOLID95 Geometry

SOLID95 Input Data

The geometry, node locations, and the coordinate system for this element are shown in Figure 1: "SOLID95 Geometry". A prism-shaped element may be formed by defining the same

node numbers for nodes K, L, and S; nodes A and B; and nodes O, P, and W. A tetrahedral-shaped element and a pyramid-shaped element may also be formed as shown in Figure 1: "SOLID95 Geometry". A similar, but 10-node tetrahedron, element is SOLID92.

Besides the nodes, the element input data includes the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems.

Element loads are described in Node and Element Loads. Pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure 1: "SOLID95 Geometry". Positive pressures act into the element. Temperatures may be input as element body loads at the nodes. The node I temperature T (I) defaults to TUNIF. If all other temperatures are unspecified, they default to T(I). If all corner node temperatures are specified, each midsize node temperature defaults to the average temperature of its adjacent corner nodes. For any other input temperature pattern, unspecified temperatures default to TUNIF.

Material properties are oriented the same way as for a shell element (using the plane through the midsize nodes Y-Z-A-B) when you set KEYOPT (1) = 1. The element z-axis is normal to this plane and the element x-axis is determined by projecting the x-axis (set with ESY) onto the midsize node plane. If needed, the x-axis can be adjusted by using THETA, an optional real constant. THETA cannot be changed between load steps.

A lumped mass matrix formulation, which may be useful for certain analyses, may be obtained with LUMPM. While the consistent matrix gives good results for most applications, the lumped matrix may give better results with reduced analyses using Guyana reduction. The KEYOPT (5) and (6) parameters provide various element printout options (see Element Solution).

You can apply an initial stress state to this element through the ISTRESS or ISFILE command. For more information, see Initial Stress Loading in the ANSYS Basic Analysis Guide. Alternately, you can set KEYOPT (9) = 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the Guide to ANSYS User Programmable Features.

You can include the effects of pressure load stiffness using SOLCONTROL,,,INCP. If an unsymmetrical matrix is needed for pressure load stiffness effects, use NROPT.UNSYM.

A summary of the element input is given in "SOLID95 Input Summary". A general description of element input is given in Element Input.

SOLID95 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom

UX, UY, UZ Real Constants

TFIETA - x-axis adjustment (used only when KEYOPT (1) =1)

Material Properties

EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ or THSX, THSY, THSZ), PRXY, PRYZ, PRXZ (or NUXY, NUYZ,

NUXZ), DENS, GXY, GYZ, GXZ, DAMP Surface Loads
Pressures —
Face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N), face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)
Body Loads
Temperatures —
T (I), T(J),T(Z), T(A), T(B) Special Features
Plasticity
Creep
Swelling
Stress stiffening
Large deflection
Large strain
Birth and death
Adaptive descent
Initial stress import
KEYOPT (1)
Element coordinate system:
0--
(Default)
1 --
Orient material properties using plane created by midsize nodes T-
Z-A-B) with the z-axis normal to that plane and the x-axis (from ESYS) projected onto that plane.
KEYOPT (5)
Extra element output: 0 --
Basic element printout 1 --
Repeat basic solution for all integration points 2--
Nodal stress printout KEYOPT (6)
Extra surface output: 0--
Basic element printout 1 --
Surface printout for face I-J-N-M 2--
Surface printout for face I-J-N-M and face K-L-P-O (Surface Print out valid for linear materials only)3--
Nonlinear printout at each integration point 4--
Surface printout for faces with nonzero pressure KEYOPT (9)
Initial stress subroutine option (available only through direct input of the KEYOPT command):0--
No user subroutine to provide initial stress (default) 1 --
Read initial stress data from user subroutine USTRESS (see the *Guide to ANSYS User Programmable Features* for user written subroutines)
KEYOPT (11)
Integration rule: 0--
No reduced integration (default) 1 --
2x2x2 reduced integration option for brick shape

PREPROCESSING STAGE

1. IMPORTING THE MODEL:

Utility menu>file>import>IGES>browse the model from specified location (which is already drawn in PRO/E and converted in to IGES format).

2. DEFINE THE TYPE OF ELEMENT

Preprocessor>element type> add/edit/delete...

For this problem we will use the SOLID 95 (structural, solid, 20 node triangle) element.

3. DEFINE ELEMENT MATERIAL PROPERTIES

Preprocessor>material props>material

models>structural>linear>elastic> Isotropic.

In the window that appears, enter the following geometric properties for steel:

i. young's modulus

EX:210000

ii. poissons ratio PRXY:0.3

4. DEFINE MESH SIZE

Preprocessor>mesh tool>the following window will appear>mesh>pick all>done>close the appeared box.

SOLUTION STAGE

1. DEFINE ANALYSIS TYPE:

Solution > analysis type > new analysis > static

2. APPLY CONSTRAINTS

Solution > define loads >apply > structural > displacement > on areas>select areas>then arrest all DOF'S

3. APPLY LOADS

Solution>define loads>apply>structural>loads >on nodes>select the correct nodes and apply the correct nodes.

4. SOLVE THE SYSTEM

Solution>solve>current LS

Solve

RESULTS AND DISCUSSION

Here we are going to compare the conventional (computation based on the standard formulae with the help of computer C-program) and ANSYS (computation using ANSYS 8.0 version) results.

The results are:

Conventional bending stress : 3.72Mpa

ANSYS bending stress : 3.55IMpa

Thus the bending stress computed is within the limit so the design is safe. Thus the bending stress computed is within the limit so the design is safe.

CONCLUSION

Thus the stress analysis on turbine shaft is made and the bending stress is computed by both conventionally and using ANSYS By comparing both the results there exit a 4.5% of error and the bending stress computed is with in the allowable limit.

REFERENCES

- [1] Shigley, Mischke, Budynans and Nisbett "Mechanical Engineering design" 8th Edition.
- [2] G.Chaitanya, J.Suresh Kumar, Kolla Srinivas "Optimization of Axial Compressor stage using NSGA-II technique", ARPN Journal of Engineering and Applied Sciences, Vol5, No 12, pages 1-5,2010.
- [3] S.S.Rao,"The Finite Element method in Engg", BH Publications New Delhi, 3rd Edition, 1999.
- [4] C.S.Krishnamoorthy, "Finite Element Analysis, Theory and Programming, 2ndedition, Tata McGraw Hill publishing company Ltd. New Delhi, 2002.
- [5] Jean-Claude Sabonnadiere and Jean-Louis Coulomb," Finite Element method in CAD", North Oxford University, 1987.