

“Analysis Of Vibration Characteristics For Annular Disc Cutters”

Pathak L. M.¹, Desavale R. G.², Kurane S.S.³

¹(P.G. Student, Department of mechanical Engineering, Annasaheb Dange College of Engineering and Technology, Ashta)

² (Asso. Prof. Department of mechanical Engineering, Annasaheb Dange College of Engineering and Technology, Ashta)

³(Assi. Prof. Department of mechanical Engineering, Ashokrao Mane Group of Institutes, Vathar.)

ABSTRACT : Circular annular cutters with uniform radial cracks are extensively used in the cutting processes. Unwanted noise, vibration & accidental failure associated with the cutting process have become an important economic and technological problem in the industry. The knowledge of natural frequencies of components is of great interest in the study of response of structures to various excitations. Hence, it is important to study a circular plate cutter with central hole, which fixed at inner edge and free at outer edge with its dynamic response. In this study some efforts are taken for analyzing vibration characteristics of annular disk cutter. From the experimentation and validation of results, it is found that this study is needful for design data preparation. Secondarily, modal analysis is one of the better tools for finding vibration characteristics for such components.

KEYWORDS : cutters, noise, vibration, accidental failure, vibration characteristics.

I. INTRODUCTION

The study of the dynamic behavior of circular cutters with free boundary condition but having different numbers of cutting teeth's, aspect ratio, effect of radial slots, enlargement of stress concentration holes is important, as used in several cutting processes. The knowledge of natural frequencies of component is of great interest in the analysis of response of structures to various excitations [1]. Unwanted noise, vibration & accidental failure associated with the cutting process have become an important economic and technological problem in the industry. Hence a circular plate cutter with central hole, fixed at inner edge and free at outer edge is chosen and its dynamic response is investigated in this paper [3, 4, 5]

II. PHASES OF THE PAPER WORK

The paper work has divided by different phases such as literature survey, theoretical investigation, experimental procedure and validation of results. The details of these phases are:

Phase I – In this phase the literatures required for annular disk cutters, vibration characteristics, methods of analysis has been studied. Secondarily, suggestions and conclusions by various researches and authors related with vibration characteristics of annular cutter discs have referred.

Phase II – Theoretical investigation

Analysis of vibration characteristics of circular cutters with free boundary condition but having different numbers of cutting teeth, aspect ratio, effect of radial slots, and enlargement of stress concentration holes is done with the FEM analysis by purchasing such cutters from market, drawing & meshing. Finite Element Method (FEM) software package is used for vibration analysis of cutters with same boundary condition for determining different parameters like natural frequency and mode shapes. Software analysis of different types of cutters through ANSYS. This will be needful in FEA for determining natural frequencies & mode shapes.

Phase III - Experimental work will carry out by using FFT analyzer Experimentation will done by using FFT analyzer. The natural frequencies will detect by hitting the plates with impact hammer and the response at a point of a plate will measure by an accelerometer. Hence, FFT analyzer can analyze the output of accelerometer [2].

Phase IV – Analysis will done by using exciter.

When the cutter will mount on exciter, then it will detect different resonance by varying exciting frequency. The various mode shapes will gain by see source, sand, coal dust or salts for comparing FEA mode shapes.

Phase V – Validate the FEA result with experimental results

III. THEORETICAL INVESTIGATION

3.1 Investigation by finite Element Method

In this method, the actual structure is replaced by several pieces or elements, each of which is assumed to behave as a continuous structural member called a finite element. The elements are assumed to be inter connected at certain point known as joints or nodes. Since it is very difficult to find the exact solution of the original structure under the specified loads, a convenient approximate solution is assumed in each finite element. During the solution process, the equilibrium of forces at the joints and the compatibility of displacement between the elements satisfied, so that the entire model is made to behave as a single entity. The finite element method is a numerical method which can be used for the accurate solution of complex mechanical and structural vibration problems. The analysis capabilities of ANSYS include the ability to solve static and dynamic structural analysis, static or time varying magnetic analysis and various types of field and coupled field applications. This program contains many special features, which allow nonlinearities or secondary effect to be include in the solutions, such as plasticity, large strain, hyper elasticity, creep, swelling, large deflection, contact stress, stiffening, temperature dependency, material anisotropy and radiation[6-7].

3.2 Modal analysis

Same set of command is used for modal analysis that used in any other type of finite element analysis. Likewise, choose similar option from the graphical user interface (GUI) to build and solve models. Modal analysis determines the vibration characteristics (natural frequencies and mode shapes) of a structure or machine components.

3.2.1. Steps of modal analysis

The various steps of model analysis are as bellow:

- Model meshing
- This step includes the job name and analysis title and then uses PRER7 to define the element types, element real constant, material properties, and the model geometry.
- Loadings
- In this step, define analysis types and options, apply load, satisfy load step options to get the finite element solution for the natural frequency.
- Expansion of the modes
- Specify the number of mode that, you have to expand. If frequency range is selected, only modes within that range mode results are appeared.
- Results

Results from modal analysis are written to the structural results file. Results consist of natural frequency, mode shapes relative stress and force distribution. Those results wish to see, database must contain the same modal for which the solution was calculated.

3.3 Modal analysis of circular cutters.

Analysis of vibration characteristics of circular cutters with free boundary condition but having different (number of cutting teeth, aspect ratio, effect of radial slots, enlargement of stress concentration holes) is done here. Analysis for circular cutters with inner edge clamped and outer edge free is done here. For same aspect ratio of annular disc but variable numbers and variable lengths of radial cracks for inner edge clamped and outer edges free boundary condition. When the excitation frequency of any structure matches with one of the natural frequency of the plate then resonance occurs and a damage of plate takes place. So here objective is to determine the modal parameters and an effort can be made to avoid the resonance

3.3.1. Analysis procedure

3.3.1.1. Specifying a job name and set the analysis title

- Choose menu path UTILITY MENU > FILE > CHANGE TITLE.
- Enter text “Modal analysis of annular disc with radial cracks” and click on OK
- Choose menu path MAIN MENU > PREFERANCES.
- Click on structural option. Click on OK.

3.3.1.2. Define element types

Choose menu path MAIN MENU > PREPROCESSOR > ELEMENT TYPE > ADD / EDIT / DELETE. The element types dialog box appears.

- Click on ADD. The library of element types dialog box appears. The ANSYS element library contains more than 100 different element types. Each element has a unique number and a prefix that identifies the element category. BEAM 188, BEAM 4, SOLID 96, SOLID 45, SOLID 92, SHELL 63, PLANE 77. Most ANSYS element types are structural elements, ranging from simple beam to more complex layered shells. The element types determines from following procedure.

1) The degree of freedom set (related to structural, thermal, magnetic, elastic, quadrilateral, brick, etc.)

2) Two-dimensional or three-dimensional element.

- In the left scroll box, click once on “structural shell”.
- In the right scroll box, click once on “Elastic 4 node 63”.
- Click on apply.
- Click on close in the element types dialog box.

3.3.1.3. Define material properties

Material properties may be orthographic or an isotropic. In this case material of the plate is mild steel having Young’s Modulus 210 MPA, Density 7850Kg/m³ and Poisson’s Ratio is 0.3 the analysis being linear modal and complete subassembly is made of the same material. It is chosen as constant and the same is entered through the menu path.

- Choose menu path MAIN MENU > PREPROCESSOR > MATERIAL PROPERTIES > CONSTANT-ISOTROPIC.
- Click on OK to specify material number 1.

A second dialog box appears. Enter values of mild steel.

3.3.1.4. Creating the model geometry

Next step in analysis is to create the geometric model and to generate a finite element mesh for the given geometry. There are two methods used to create finite element model; one is solid modeling and other is direct generation. Use solid modeling, which automatically mesh the geometry with nodes and elements (refer Fig.1). MAIN MENU > PREPROCESSOR > MODELING > CREATE > AREAS > CIRCLE > ANNULAR

- Insert accurate co-ordinate of the geometry and then click on OK.
- Insert accurate co-ordinate of the plates
- Insert dimensions of radius first and second and then click OK

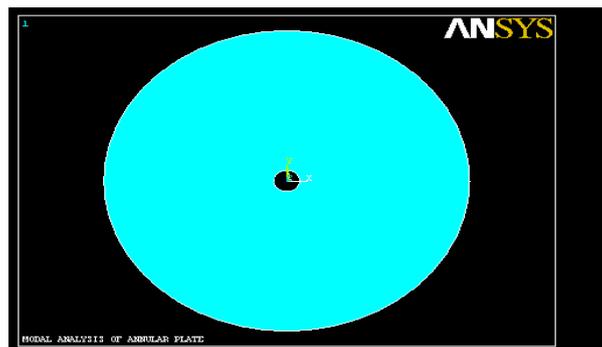


Fig. 1 Annular disk model

3.3.1.5. Mesh the model

Mesh the model with nodes and elements. Before meshing the model, even before building the model, it is important to think about whether a free mesh or a mapped mesh is used. Mapped mesh is approximate for the analysis. A free mesh has no restriction in terms of elements shapes and has no specified pattern applied to it. A mapped area mesh contains either only quadrilateral or only triangular elements, while mapped volume mesh contains only hexahedron elements. Once the plate structure model will complete the element attribute set and then ANSYS program will be directed to generate a free mesh. The technique used for meshing is free mesh with smart sizing technique and select a refinement level.

- Click on MESH TOOL.
- Select SMART SIZE. Then click on MESH.
- Select model for meshing and Click on OK.

3.3.1.6. Applying loads

It includes (constraints, supports or boundary field specification) i.e. boundary condition as well as other externally or internally applied load. Degree of freedom constraints, forces (point loads), surface loads, body loads and varying loads, inertia loads and coupled field loads. One can apply most of these loads either on the solid model or finite element model. Structural load is considered and structural zero displacement is entered and specified boundary conditions were applied.

3.3.1.7. Specifying load step options

It is a configuration of loads for which one has obtained a solution in a structural analysis. For example, one may apply wind loads in one load step and gravity in second load step. Sub steps are incremental steps taken within a load step. One uses them mainly for accuracy and convergence purpose in transient and non-linear analysis. Sub-steps are also known as time steps. Depending upon the type of analysis is doing, load step option may or may not be required. For the plate analysis it was not necessary to satisfy the load step as it involved only steady state loading, the load step was not entered, ANSYS program takes it as by default.

3.3.1.8. Specify analysis type and option

- Choose menu path MAIN MENU> SOLUTION > ANALYSIS > TYPE > NEW ANALYSIS
- Click on model analysis option and click OK
- Choose menu path MAIN MENU > SOLUTION >ANALYSIS OPTION. The model analysis dialog box appears.
- Click on subspace option on. Enter 10 for number of modes to extract

3.3.1.9. Specify number of modes to be expanded

- Choose menu path MAIN MENU > SOLUTION > LOAD STEP >OPTION >EXPANSION PASS > EXPAND MODES. Then expand modes dialog box appears.
- Enter 10 numbers of modes to expand and click on OK.

3.3.1.10. Initialing the solution

- Choose path MAIN MENU > SOLUTION > SOLVE >CURRENT LS
- When this command is issued to the ANSYS program ,which takes model and loading results provided for the plate structure from the database and calculates the results.

3.3.1.11. Review of obtained results

Results are reviewed in general post-processor and time history in post-processor. The general postprocessor is used to review the results during a particular time step for the whole or part of body. Different results like deformations, stresses, and tabular listings of displacements and animations of vibration can be obtained by using general post-processor. Time-history postprocessor is used to find out results of a particular node through all the time-steps. By using time-history postprocessor, graphs of data vs. time can be obtained. The arithmetic operations can also be performed on graphs.

IV. EXPERIMENTAL WORK.

Experimental modal analysis of a system, deals with determination of natural frequencies, damping ratios, and mode shapes through the vibration testing [5]. In the case of forced vibration, the analysis includes the study of acceleration, velocity and displacement responses of the systems. The schematic setup for conducting experiment has shown in Fig. 2.

4.1 .Basic assumption in experimental analysis

Four basic assumptions are used to perform an experimental modal analysis.

- The structure is assumed to be linear i.e. the response of the structure to any combination of forces, simultaneously applied, is the sum of the individual responses to each of the forces acting alone.
- The structure is time invariant, i.e. the parameters that are to be determined are constants. In general, a system which is time invariant has components whose mass, stiffness, or damping depend on factors that are not measured or not included in the model.

- The structure obeys Maxwell's reciprocity, i. e. a force applied at degree of freedom p causes a response at degree of freedom q that is the same as the response at degree of freedom p caused by the same force applied at degree of freedom q.
- The structure is observable; i. e. the input output measurements that are made enough information to generate an adequate behavioral model of the structure

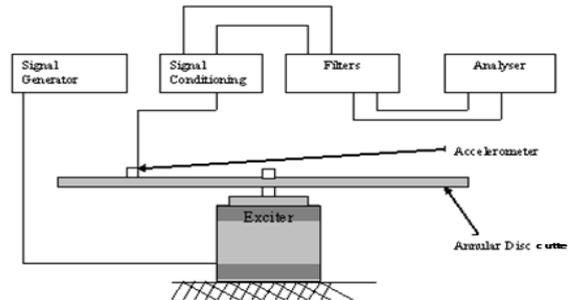


FIG. 2 EXPERIMENTAL SET UP

4.2 Instrumentation used for modal analysis

- FFT analyzer
- Accelerometer
- Exciter
- Impact hammer

4.2.1 Fast fourier transform

A spectrum analyzer is an electronic device that is capable of taking the time waveform of a given signal and converting it into its frequency domain. Importance of spectrum analyzer by J. B. Fourier mathematician showed that it is possible to represent any time waveform (the plot of a signal whose amplitude varies with time) by a series of sins and cosines of particular frequencies and amplitudes.

4.2.1.1 Two channel spectrum analyzer

A Two - channel spectrum analyzer is far more powerful than signal channel analyzer. The two-channel analyzer operates in same way as a single channel analyzer with following exception:

- Two input attenuators.
- Two input buffers, controlled by the same analyzer in the internal clock.
- Half the number of lines of resolution as the same, analyzer in the single – channel mode.
- Calculation of cross channel properties such as the transfer function, coherence and coherent –output power.

4.2.2. Accelerometer

The benefit of use of accelerometers is that they do not require a calibration program to ensure accuracy. From the accelerometer record, the velocity and displacement are obtained. A wax type material is used to mount the accelerometer on the non-magnetic material. It is a linear seismic transducer utilizing a piezoelectric element in such a way that an electric charge is proportional to the applied acceleration Piezoelectric accelerometers utilize a variety of seismic element configurations. Most are constructed of polycrystalline ceramic piezoelectric materials because of their ease of manufacture, high piezoelectric sensitivity, and excellent time and temperature stability.

4.2.3. Vibration exciter

Vibration exciter SI-220 is an electrodynamic type of device. By controlling the amount of current, the amplitude of vibration is controlled. A powerful magnet placed centrally surrounding which is suspended with the exciter coil. This assembly is enclosed by a high permeability magnetic circuit for optimum performance and enough design care has been observed to minimize the leakage magnetic flux at the top of the vibration table. When an electrical current is passed through the exciter coil, a magnetic field is created around the coil. This field interacts with the field due to the central permanent magnet and these results in the upward or downward movement of the suspended coil depending upon the direction of current flow in the coil. If an alternating

current is injected into the coil, it moves up and down continuously. Thus controlling the frequency of the coil current, the frequency of vibration is controlled.

4.2.4. Impact hammer

It is a built up force transducer in its head. The impact hammer can be used to hit or impact the structure being tested to excite a wide range of frequencies. The shape of frequency response is dependent on the mass and stiffness of the hammer and structure. The impact force caused by impact hammer, which is nearly proportional to the mass of the hammer head and the impact velocity. It can be found from the force transducer embedded in the structure at an impulse is composed of excitation at each of the natural frequencies of the structure. The impact hammer is simple portable, inexpensive and much faster to use than exciter.

V. VIBRATION MEASURING TECHNIQUES

5.1 Time domain analysis

It uses the history of the signal (waveform). The signal is stored in an oscilloscope or a real-time analyzer and any non-steady or transient impulse are noted.

5.2 Frequency domain analysis

Frequency spectrum/domain is a plot of the amplitude of vibration response versus the frequency and can be derived by using the digital fast fourier analysis of the time waveform. Another important characteristic of a spectrum is that each rotating element in a machine generates identifiable frequency. The frequency spectrum provides valuable information about the condition of the machine. The vibration of response of a machine is governed not only by its component but also by its assembly, mounting and installation. Thus the vibration characteristics of any machine are somewhat unique to that of particular machine.

VI. MEASUREMENT METHODS

Measurement methods are used to collect data from the tested structure, i.e. to obtain the various mobility properties in the form of a frequency response function. To be able to describe or simulate an existing system accurately, high quality measured data is required. Depending on the type of structure, the time available to perform the tests among others, several methods can be applied for excitation. Some aspects of the measurement process which require particular attention are:

- Mechanical aspects of supporting a structure.
- Mechanical aspects of exciting a structure.
- Correct transduction of the quantities to be measured by the transducers (force, displacement, motion and acceleration).
- Appropriate signal processing.

6.1 Exciting the structure

There are several methods allowing to do so, each having its particular characteristics and applications, depending on the type of system analyzed, the quality of the data that is required, the time that is available for the measurements for instance.

6.1.1. Harmonic excitation

This frequency response function is obtained by using steady-state harmonic excitation. Sine excitation is among the most commonly applied excitation methods to obtain frequency response functions.

6.1.2. Random excitation

It is used in linear structural dynamic tests because the characteristics of random signals of containing energy over a wide range of frequencies, simultaneously. This method requires never-repeating and infinite signals

6.1.3. Transient excitation

To perform this kind of testing, we excite the structure by means of a hammer blow and measure the response of the structure. As an impact excitation, a widespread and convenient excitation technique. Above two methods does not require a permanently attached device to produce the force signal on the structure. To perform this kind of testing, we need a shaker attached to the structure like in the case of sine excitation.

VII. VIBRATION MEASUREMENT SYSTEM

Feature of the measurement scheme is shown in the Fig.3. Here motion of the vibrating body is converted into an electrical signal by the vibration transducer. In general a transducer is a device that transforms the signal changes in mechanical quantities (such as displacement, velocity, acceleration, force) changes into electrical quantities (such as voltage, current). Since the output signal conversion instrument is used to amplify the required value. The output from the signal conversion instrument can be presented on display unit for visual inspection, or by recorder by recording device or stored in a computer for later use. Depending upon the quantity measured a vibration measuring instrument is called a vibrometer, a velocity meter, an accelerometer, a phase meter, or a frequency meter. The following consideration often dictates the type of measuring instrument to be used in a vibration test

- Size of machine/structure involved.
- Expected range of frequencies and amplitude.
- Condition of operation of the machine/equipment/structure.
- Type of data processing used (such as graphical display or graphical recording or storing the record in signal form for computer processing).

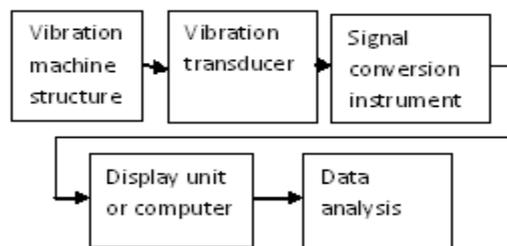


FIG.3 VIBRATION MEASUREMENT SYSTEM

VIII. CONCLUSIONS AND DISCUSSION

- This study is useful as fundamental design data preparation for high risk application (like circular saw cutters, flywheel). Because change in above variable like natural frequency and mode shapes will be change accordingly.
- Model analysis is the identification of vibration characteristics of elastic structures. It consist of describing a system by its modal parameters like natural modes. So this will be also helpful for design changes in future as design data

REFERENCES

- [1] Weisensel G. N., "Natural Frequency information for circular and annular Plates", Journal of sound and vibration, vol. 133, pp129-134, 1989.
- [2] D.V. Bambill, S. La. Malfa, C. A. Rossit, P. A. A. Laura, "Analytical and Experimental investigation on transverse vibration of solid, circular and Annular plates carrying a concentrated mass at an arbitrary position with Marine applications", Journal of ocean Engineering, vol31, pp127-138.
- [3] J. C. Bae and J. A. Wickert, "free vibration of coupled disk-hat Structures", Journal of sound and vibration, 2000, 235(1), 117-132.
- [4] Vogel S. M. & Skinner D.W., "Natural Frequencies of Transversely Vibrating Annular Plates", Journal of Applied Mechanics, December, 1965, 926-931.
- [5] K. Ramesh, D. P. S. Chauhan and A.K. Mallik, "Free vibration of annular plate with periodic radial cracks", Journal of sound and vibration, 1997,206(2)266- 274.
- [6] M. J. Fagan, finite element analysis-theory and practice (Longman Scientific and technical)
- [7] U. S. Dixit, finite element method for engineers (Cengage learning 2002)