

Model Analysis of Axially Symmetric Linear Rotating Structures

M. Varun Kumar, B. Ashwini Kumar

Abstract- Rotor dynamics is the study of vibrational behavior in axially symmetric linear rotating structures. Devices such as engines, motors, disk drives and turbines all develop characteristic inertia effects that can be analyzed to improve the design and decrease the possibility of failure. At higher rotational speeds, the inertia effects of the rotating parts must be consistently represented in order to accurately predict the rotor behavior and to decrease possibility of failure.

In this mini project we are going to design a rotor model using finite element analysis (FEA) in ANSYS as per standard dimensions. After modeling in ANSYS we are going to perform the modal analysis in ANSYS using commands by using sub structuring and super element generation.

If the frequency of any harmonic component of a periodic phenomenon is equal to (or) approximates the natural frequency of any mode of rotor vibration, a condition of resonance may exist. If resonance exists at a finite speed, that speed is called a critical speed. An important part of the inertia effects is the gyroscopic moment introduced by the precession motion of the vibrating rotor as it spins. This gyroscopic is accounted in the modal analysis in ANSYS.

Keywords-- Vibrations, Bearings, gyroscopic effect, Campbell diagram

I. INTRODUCTION

Rotor dynamics is a specialized branch of applied mechanics concerned with the behavior and diagnosis of rotating structures. It is commonly used to analyze the behavior of structures ranging from jet engines and steam turbines to auto engines and computer disk storage. At its most basic level rotor dynamics is concerned with one or more mechanical structures (rotor) supported by bearings and influenced by internal phenomena that rotate around a single axis. The supporting structure is called a stator. As the speed of rotation increases the amplitude of vibration often passes through a maximum that is called a critical speed. This amplitude is commonly excited by Unbalance of the rotating structure; everyday examples include engine balance and tire balance. If the amplitude of vibration at these critical speeds is excessive then catastrophic failure occurs. In addition to this, turbo machinery often develops instabilities which are related to the internal makeup of turbo machinery, and which must be corrected. This is the chief concern of engineers who design large rotors.

An important part of the inertia effects is the gyroscopic moment introduced by the precession motion of the vibrating rotor as it spins. As spin velocity increases, the gyroscopic moment acting on the rotor becomes critically significant. Not accounting for these effects at the design level can lead to bearing and/or support structure damage.

Accounting for bearing stiffness and support structure flexibility, and then understanding the resulting damping behavior is an important factor in enhancing the stability of a vibrating rotor. Rotor dynamics can be divided into three different types of motion, lateral, longitudinal and Torsional. Lateral is also called bend rotor dynamics and is associated with bending of the rotor. Torsional is the modes when the rotor is twisting around its own axis. Longitudinal modes are when the rotor parts are moving in axial direction. Every system has its own natural frequencies and if the disturbing force's frequency is close to any of the natural frequencies, the amplitude can get very large. This phenomenon is called resonance. To hold the amplitudes on a decent level, damping can be applied to the system. Damping is for example applied with help of the Bearings. When dealing with torsional motion the whole system together with the generator has to be modeled even if there is a gearing between them because the twisting motions are affecting each other. The whole system doesn't have to be modeled when lateral and longitudinal modes should be analyzed. Lateral vibrations, torsional vibrations and longitudinal vibrations cannot always be calculated separately from each other because sometimes they are affecting each other.

II. DIFFERENT TYPES OF VIBRATIONS ARE

A. Torsional Vibrations

Torsional vibration is when the oscillation motion is twisting the rotor. The oscillations are added to the constant rotational speed of the rotor. When a rotor should be designed with respect to torsional vibrations there are four important analyses which have to be done, static, real frequencies, harmonic force response and transient. In this thesis work only real frequencies and harmonic force response has been focused on. Static analysis is easily done by just calculating the stresses when the rotor is held fixed in one end and twisted by a torque in the other end. In real frequencies analysis the natural frequencies are evaluated in order to decide the critical speeds of the rotor. Harmonic force response analysis is done to see how large the twisting motion on the rotor is when the rotational speed is close to any of critical speeds. In a transient analysis the response of the rotor is calculated after a large torque has been affecting the rotor for a short while. The torque can come from a generator during a power failure which creates the large torque. The analysis is focused on to see what happens after the torque is released and to see how long time it takes to reduce the oscillations in the rotor. There is almost no damping in a torsional motion, the only damping is material damping which is small compared to the bearing damping.

Manuscript received on April, 2013.

M. Varun Kumar, Department of Mechanical Engineering, KL University, Vijayawada, INDIA

B. Ashwini Kumar, Department of Mechanical Engineering, Assistant Professor, KL University, Vijayawada, INDIA

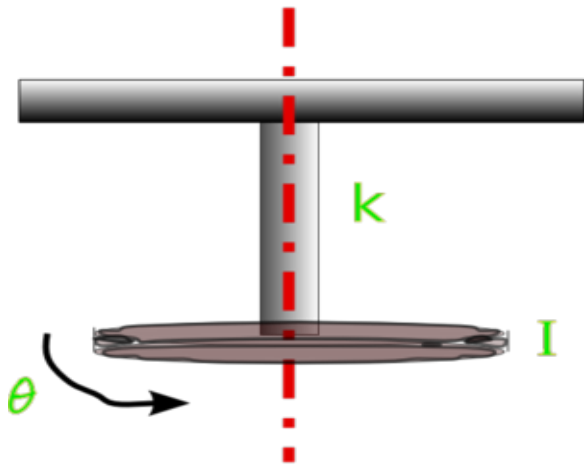


Fig.1, torsional vibrations

B Longitudinal Vibrations

Longitudinal vibrations consist of an extension and compression motion of the rotor. The motion can be simplified with two masses with a spring between them. The masses are then moving back and forth from each other. To calculate longitudinal vibrations are similar to calculate torsional vibrations except for using the mass instead of the polar mass moment of inertia. The damping for longitudinal motion is almost zero, only material damping occurs which is small.

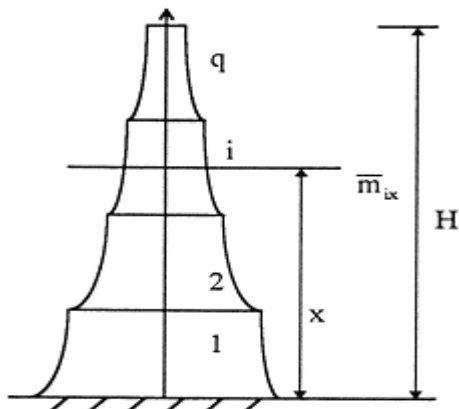


Fig.2, longitudinal vibrations

C. Lateral vibration

Calculations of lateral vibrations are more complex than torsional vibrations and more analyses have to be done. The analyses which have to be done are static, harmonic force response, real frequencies and complex Eigen values. Stator-simulations and bearing calculations have to be done also because they are affecting the lateral vibrations. The static analysis is done to see how much the rotor is deflecting by its own weight. Harmonic force response analysis is done to see how much the rotor is deflecting when the rotational speed is close to any critical speed. It is important to calculate the deflection because the rotor is never allowed to touch the bearings. Real frequencies analyses are done get the real frequencies, which are the frequencies without damping. It's also called undamped critical speeds analysis, Complex eigenvalues analyses are done to get the critical speeds and that is done by setting up a Campbell diagram. The analysis is also called damped critical speeds, Bearing calculations need to be done to know how stiff the bearings are and how much damping the bearings give. This is done by a special program constructed to calculate these properties.

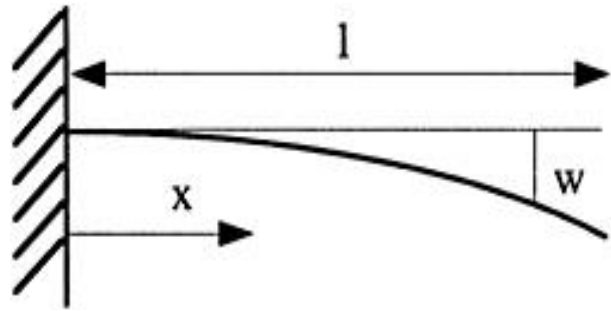


Fig3, lateral vibrations

III. PARTS OF A ROTATING STRUCTURE

A. Stator

The stator is everything which is placed around the rotor, like the housing and cables. The stator and the rotor are held together with oil bearings (more about bearings below).The rotor-stator interaction is a predominant part of the noise emission.

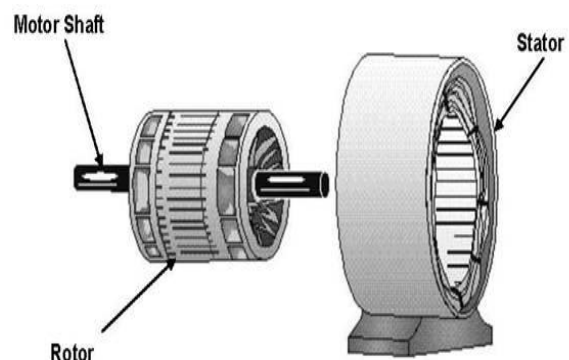


Fig4, stator part

B. Bearings

The bearings are supporting and holding the rotor in the right position in the stator. It is very important that the bearings are properly constructed because they influence the rotor dynamics and they need to have a long lifetime. Some factors which influence the bearing design are.....

- Load and rotational speed of the rotor
- Lubrication of the oil
- Temperature of the oil
- Shaft arrangements
- The life of the bearings
- Mounting and dismantling
- Noise
- Environmental condition.

The best one is tilting-padding bearing because it has the best stability properties. The best thing is that the cross coupling springs can be ignored if it is modeled by four springs. This means that the bearing can be modeled by one spring in vertical direction and one spring in horizontal direction. The problem is that stability decreases the efficiency and a tilting padding bearing cost a lot more than the rest of the bearings. The oil consumption is higher on the bearing with better stability because of thicker oil film is needed.

IV. HISTORY

The history of rotor dynamics is replete with the interplay of theory and practice. W. J. M. Rankine first performed an analysis of a spinning shaft in 1869, but his model was not

adequate and he predicted that supercritical speeds could not be attained. In 1895 Dunkerley published an experimental paper describing supercritical speeds. Gustaf de Laval, a Swedish engineer, ran a steam turbine to supercritical speeds in 1889, and Kerr published a paper showing experimental evidence of a second critical speed in 1916.

Henry Jeffcott was commissioned by the Royal Society of London to resolve the conflict between theory and practice. He published a paper now considered classic in the Philosophical Magazine in 1919 in which he confirmed the existence of stable supercritical speeds. August Foppl published much the same conclusions in 1895, but history largely ignored his work.

Between the work of Jeffcott and the start of World War II there was much work in the area of instabilities and modeling techniques culminating in the work of Prohl and Myklestad which led to the Transfer Matrix Method (TMM) for analyzing rotors. The most prevalent method used today for rotor dynamics analysis is the Finite Element Method.

Modern computer models have been commented on in a quote attributed to Dara Childs, "the quality of predictions from a computer code has more to do with the soundness of the basic model and the physical insight of the analyst. ... Superior algorithms or computer codes will not cure bad models or a lack of engineering judgment."

Prof. F. Nelson has written extensively on the history of rotor dynamics and most of this section is based on his work.

There are many software packages that are capable of solving the rotor dynamic system of equations. Rotor dynamic specific codes are more versatile for design purposes. These codes make it easy to add bearing coefficients, side loads, and many other items only a rotor dynamics would need. The non-rotor dynamic specific codes are full featured FEA solvers, and have many years of development in their solving techniques. The non-rotor dynamic specific codes can also be used to calibrate a code designed for rotor dynamics.

V. THE GENERAL DYNAMIC EQUATIONS

The general dynamic equation is

$$[M]\{\ddot{U}\}+[C]\{\dot{U}\}+[K]\{U\} = \{f\}$$

$[M]$, $[C]$, $[K]$ are mass, damping, stiffness matrices

In rotor dynamics, equation gets additional contributions from gyroscopic effects $[G]$, rotating damping effect $[B]$ i.e.

$$[M]\{\ddot{U}\}+[G]+[C]\{\dot{U}\}+[B]+[K]\{U\}=\{f\}$$

In this modal analysis we does not consider damping and other functions, so equation is

$$[M]\{\ddot{U}\}+[K]\{U\} = 0$$

The gyroscopic matrix $[G]$ depends on rotational velocity

The damping effect $[B]$ also depends upon rotational velocity. It modifies the apparent stiffness of the structure and can produce unstable motion.

The gyroscopic matrix G is proportional to spin speed Ω . The general solution to the above equation involves complex eigenvector which are spin speed dependent. Engineering specialists in this field rely on the Campbell Diagram to explore these solutions.

An interesting feature of the rotor dynamic system of

equations is the off-diagonal terms of stiffness, damping, and mass. These terms are called cross-coupled stiffness, cross-coupled damping, and cross-coupled mass. When there is a positive cross-coupled stiffness, a deflection will cause a reaction force opposite the direction of deflection to react the load, and also a reaction force in the direction of positive whirl. If this force is large enough compared with the available direct damping and stiffness, the rotor will be unstable. When a rotor is unstable it will typically require immediate shutdown of the machine to avoid catastrophic failure.

VI. Terminology used in Rotor dynamics are

- a) Gyroscopic effect
- b) Whirl
- c) Stability
- d) orbit
- e) Critical Speed
- f) Damping
- g) Mode shapes

A. Gyroscopic effect

For a structure spinning about an axis β , if a rotation about an axis perpendicular to β (precession motion) is applied to the structure, a reaction moment appears, that reaction is the gyroscopic moment. It axis is perpendicular to both spin axis and precession axis. One of the effects that separate vibrations of a rotor from other vibrations is the influence of gyroscopic stiffening effect. The influence of gyroscopic stiffening effects only shown in rotational parts, for example a rotor. The gyroscopic stiffening effect has everybody felt, if you are holding a rotating wheel at axis of rotation. If the wheel is held still you will not feel any forces, but if the wheel starts to rotate transversal a moment will be produced which is trying to rotate it back to the original position and this is the gyroscopic stiffening effect. This effect can be seen in forward whirling, when a moment is affecting the disk so the rotor gets stiffer and the natural frequencies will increase. It is the opposite effect on back whirling where the gyroscopic stiffening effect produces a moment that is opposite directed so the structure gets weaker. Where the gyroscopic effects are coming from will be explained by a simple model

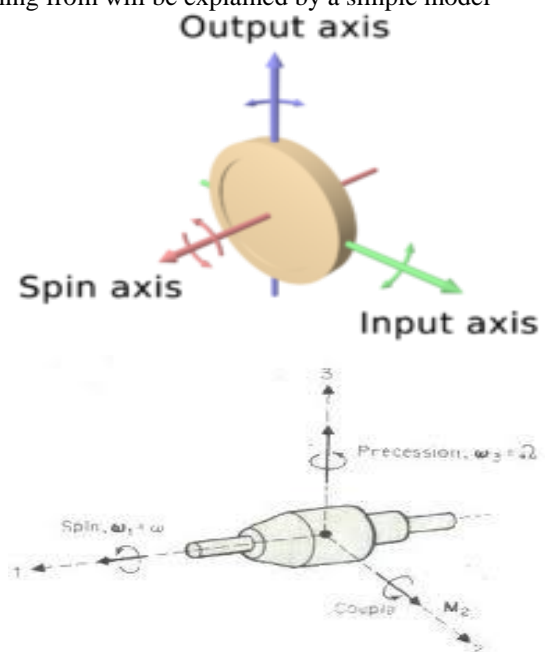


Fig5, gyroscopic effect

B. Whirl

When a rotor is operating it is not standing still at the axis of rotation; it is moving in a circular or elliptical motion around the axis of rotation. The motion is called whirling. One thing that produces the whirling motion is the centrifugal forces which make the rotor bend. Another thing can be that the rotor is not totally axis-symmetric. There are two types of whirling, forward and backward. Forward whirling is when the whirling rotation is the same direction as the rotational speed. This type of whirling is the most dangerous one because it is easier to excite the rotor with forward whirling in resonance than with backward whirling. Backward whirling is when the whirling rotation is opposite the rotational speed. Where a forward whirling is increasing the natural frequency with higher rotational speed and backward whirling is decreasing the natural frequency with higher rotational speed.

C. Stability

Self-excited vibrations in a rotating structure cause an increase of the vibration amplitude over time.

We have measurement called the log decrement (δ); can be used to decide the stability and that is a better measurement because it is a non-dimensional quantity. That makes it easier to compare it with other machines. The rotor is stable if the log decrement is positive and it is calculated from the real and imaginary values from the modal analysis. The log decrement is measuring how fast a free vibration decrease which is the vibrations that are not affected by any external forces.

Common source for Instabilities are

- Internal rotating damping.
- Contact between rotor and stator parts.
- Bearing characteristics.

D. Orbits

In most general cases, the steady state trajectory of a node located on spin axis also called as orbit.

E. Critical Speed

A critical speed appears when the natural frequency is equal to the excitation frequency. The excitation may come from unbalance which is synchronous with the rotational velocity or from any asynchronous.

E.1. Critical speed of shafts

All rotating shaft, even in the absence of external load, deflect during rotation. The combined weight of a shaft and wheel can cause deflection that will create resonant vibration at certain speeds, known as Critical Speed. The magnitude of deflection depends upon the followings:-

- (a) Stiffness of the shaft and its support.
- (b) total mass of shaft and attached parts.
- (c) Unbalance of the mass with respect to the axis of rotation.
- (d) The amount of damping in the system Therefore, the calculation of critical speed for fan shaft is necessary. Critical speed can be calculated through Campbell diagram. Brief description about Campbell diagram

Campbell Diagram: Determination of natural frequencies and mode shapes as a function of rotor speed and plotting the Campbell diagram is also well established. In diagram drawn between speed (rpm) and natural frequency (hz).

The diagram has the rotational speed of the rotor on the x-

axis and the mode frequencies on the y-axis. The modes are plotted for different rotational speeds. The frequencies are not constant over the rotational speed range because most of the modes will be increased or decreased with higher rotational speed. Forward whirling, modes are increasing and backward whirling modes are decreasing. Torsional and longitudinal modes are constant over the rotational speed range because they are not affected by the gyroscopic effect, the bearings or the stator.

For example

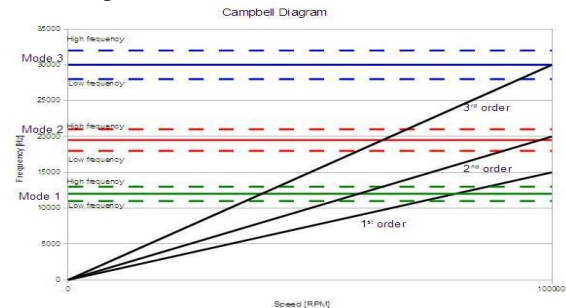


Fig6, Campbell diagram

F. Damping

Damping is the dissipation of energy which means that energy leaves the system for example as heat. There are a few different types of damping such as viscous, friction and material damping. Viscous damping can be the energy which is released when a system is moving in a fluid. Friction, described by e.g. the Coulomb law, is the damping experienced when a system for example is moving on a dry surface. Material damping is due to friction in the material itself that results in energy dissipation. A system can be over damped, under damped or critically damped. When a system is under damped it will stop in an oscillated motion and if it is over damped it will stop without any oscillations. Critical damped is the lowest damping a system can have without any oscillations. To know if the system is critically damped or over damped is important when a modal analysis should be done in Ansys.

F.1. Effects of the stiffness of the support

The supports of a rotor are always flexible because where is nothing in the real world which can be called rigid support. The stiffness of the support should be almost the same as the rotor or sometimes lower. Two reasons why the support stiffness should be lower than the rotor stiffness are because it reduces the loads which are transmitted to the stator and that will lower the structural vibration of the stator. The second reason is because the damping will be more effective which will decrease the whirl amplitude at the critical speed.

G. Mode Shapes

At the natural frequencies the motion of a system starts to move with a special pattern and all the parts in the system moves together with each other. This is called a mode. The problem when dealing with rotors is that the modes are changing with the rotational speed and they are also affected by the bearings and stator. To know all the modes are important during construction a turbine because the operating velocity cannot be close to any modes. If it is, some external forces can make the amplitude grow if the system is not adequately damped. To know the shape of the

modes are important when a rotor going to be balanced. If the engineer knows the shape he or she can much easier set out the balance weights on the balance planes on the rotor. The modes can be torsional, longitudinal or lateral.

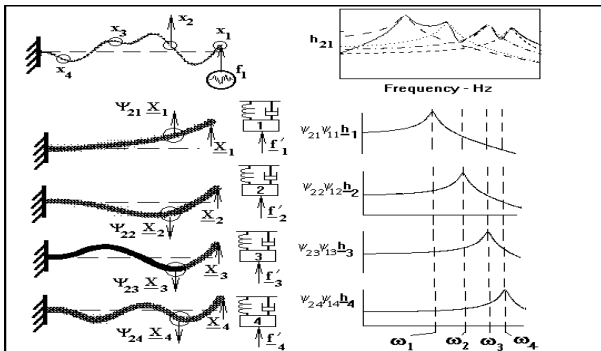


Fig7, different mode shapes of cantilever beam

VII. STATIONARY REFERENCE FRAME IN ANSYS

By using a stationary reference frame the model can have both rotating parts and stationary parts. The coordinate system which is used is the global coordinate system and that is always standing still. This is perfect when a turbine should be modeled because there are both a rotating part (rotor) and a stationary part (stator). In Ansys is a stationary reference frame activated by coriolis,on,,,on in the command window. The first "on" means that the gyroscopic effect is taken into account during the calculations and the second "on" means that a stationary reference frame will be activated. The rotating part must be axis-symmetric otherwise the gyroscopic matrix will be ignored. Ansys is recommending a stationary reference frame compared to a rotating reference frame when modal analysis should be done since by using a stationary frame Campbell diagrams can be drawn which is important when critical speeds are calculated. The gyroscopic damping matrix is only valid for linear analyses. The analyses which can be done are modal, transient and harmonic force response. Modal analyses are deciding the natural frequencies and transient analyses are done to see how long time it takes for a system to stop vibrate after it has been affected by a force. Harmonic force response analyses are done to see how the system is deflecting when it is affected by a mass unbalance force. If prestresses are accounted into the calculation, a static analysis has to be done before the modal analysis to get all the stresses and displacements in the structure. The prestresses in the model can change the Eigen frequencies.

VIII. BASIC STEPS IN ANSYS MODELING

Basic dimensions for a simple rotor and supporting structure are

Shaft length	= 595mm
Shaft diameter	= 15mm
Support block length	=790mm
Support block height	=250mm
Support block width	=154mm
Support block thickness	= 25mm
Bearing stiffness	=5e7N/m
Materials –steel – E=2e5	aluminum- E=7e4.
	$\rho=0.3$ $\rho=0.3$.

The whole text will run on input given by us.

In analysis we go through substructure analysis.

Sub structuring is a procedure that condenses a group of finite elements into one element represented as a matrix. This single matrix element is called a super element. You can use a super element in an analysis just as you would any other ANSYS element type. The only difference is that you first create the super element by performing a substructure generation analysis. Sub structuring is available in the ANSYS/Multiphysics, the ANSYS/Mechanical, and the ANSYS/Structural products.

The reasons for sub structuring are (a) to reduce computer time and (b) to allow solution of very large problems with limited computer resources. Examples of reason (a) are nonlinear analyses and analyses of structures that contain repeated geometrical patterns. In a nonlinear analysis, you can substructure the linear portion of the model so that the element matrices for that portion need not be recalculated every equilibrium iteration. In a structure with repeated patterns (such as the four legs of a table), you can generate one super element to represent the pattern and simply make copies of it at different locations, thereby saving a significant amount of computer time.

You can also use sub structuring on models with large rotations. For these models, ANSYS assumes each substructure to rotate about its mass center. In 3-D cases, the substructures contain three rigid body rotations and three translational motions. With a large rotation model, you do not constrain the substructure until the use pass because each substructure is treated as a single finite element that should allow rigid body motions.

An example of reason (b) is an analysis that is too large for the computer in terms of wave front size or disk space requirements. In such a situation, you can analyze the model in pieces, where each piece is a super element small enough to fit on the computer.

IX. ANALYSIS COMMANDS IN ANSYS

The most important commands when rotor dynamics calculations should be done

Coriolis, on

Coriolis,on is used on rotating components in a rotation reference frame. Ansys is when taking in account Coriolis matrix and spin softening matrix automatically.

Coriolis, on,,, on

Coriolis, on,,, on is used for rotating components in a stationary reference frame.

Ansys is taking in account the gyroscopic matrix in the calculation. Important is that the component is axis-symmetric around the axis of rotation.

Omega

Omega makes the whole model rotate around the global coordinate system.

Cmomega

Cmomega is the same as omega but the difference is that cmomega is pointing at a component not the whole model and is only rotating that component around a defined axis. Thanks to this the model can have many different components rotating with different rotational speed around different axis.

Pstres, on

Prestres is used when the structure is pre stressed because of temperature difference in the structure and the centrifugal force. To do an analysis, a static analysis has to be done before the modal analysis. This is because the program

needs to know all the stresses and displacements in the structure before calculating the modes.

Campbell,on

Is used when pre stressed should be account in the model. The command is saying to the program that the eigenvalues should be saved for all the sub steps.

Plcamp,, 1, rpm

Plcamp is used in the post processing to show the Campbell diagram. "1" is the slope of a line of how many excitations per revolution, thus "1" means one excitation per revolution. Rpm is the unit of the x-axis of the diagram. Rpm means revolution/min and there is also rds which means radians/second. Then cmomega is used the component name has to be written in the command also.

Prcamp,, 1, rpm

Prcamp is writing out the critical speeds from the Campbell diagram. The critical speed are there they excitation line meets any mode lines. Rpm is the unit which the rotational speed will be written out in. "1" is the slope of the excitation line which is drawn in the Campbell diagram. Then cmomega is used the component name has to be written in the command also.

Plorb

Plorb is used for showing the whirling motion of the rotating structure. The model will then be shown like rings or ellipses around the axis of rotation. Solid elements cannot show the orbits with plorb only beam and pipe elements.

Prorb

Prorb writes out the properties of the orbit motion of the rotating structure. In the program is A the major axis, B the minor axis, PSI the angle between y axis and major axis and PHI is the angle between initial position and major axis. The maximum values of Y and Z direction are also written out. This information is written out for all the nodes that are activated.

Rmore

Rmore is used when more than six constants should be added to a real constant. The extra constants will be added to the real constant which is standing above the command.

Vsum

Vsum is calculating the weight and moment of inertia of the volume that is activated.

CMS-component mode synthesis.

IX.1. Explanation of commands

The structure name is simple rotor supported by CMS super element.

Masses, polar moment of inertia, inertia are represented as m,J,I.

The abbreviations used in parameter list are used in program (as shortcuts).

CSYS: - Defines a working plane location and orientation based on an existing coordinate system.

WPOFFS:- origin of the working plane by translating the working plane along its coordinate system axes.

CSWPLA: - Defines a local coordinate system at the origin of the working plane.

VSBV: - Generates new volumes by subtracting the regions common to both NV1 and NV2 volumes (the intersection) from the NV1 volumes. The intersection can be a volume(s) or area(s). If the intersection is an area and SEPO is blank, the NV1 volume is divided at the area and the resulting volumes will be connected, sharing a common area where they touch.

NV1,NV2 are the volume which is subtract from model.

SEPO- The resulting volumes will have separate, but coincident area(s) where they touch.

VOVLAP:- Overlaps volumes. Generates new volumes which encompass the geometry of all the input volumes. The new volumes are defined by the regions of intersection of the input volumes, and by the complementary (non-intersecting) regions. This operation is only valid when the region of intersection is a volume. See the BOPTN command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

SECTYPE:- sets the section ID number, section type, and subtype for a section. If the section ID number is not specified, ANSYS increments the highest section IDnumber currently defined in the database by one. A previously-defined section with the same identification number will be redefined.

SECOFFSET:- Defines the section offset for cross sections. The location of the nodes in the section.

LESIZE:-

Defines the number of divisions and the spacing ratio on selected lines. Lines with previously specified divisions may also be changed.

ALLSEL:-It is a convenience command that allows the user to select all items of a specified entity type or to select items associated with the selected items of a higher entity.

An entity hierarchy is used to decide what entities will be available in the selection process. This hierarchy from top to bottom is as follows: volumes, areas, lines, key points, elements, and nodes. The hierarchy may also be divided into two branches: the solid model and the finite element model. The label ALL selects items based on one branch only, while BELOW uses the entire entity hierarchy. For example, ALLSEL, ALL, VOLU selects all volumes, areas, lines, and key points in the data base. ALLSEL,BELOW,AREA selects all lines belonging to the selected areas; all key points belonging to those lines; all elements belonging to those areas, lines, and key points; and all nodes belonging to those elements.

NSEL:- Selects a subset of nodes. For example, to select a new set of nodes based on node numbers 1 through 7, use NSEL,S,NODE,,1,7. The subset is used when the ALL label is entered (or implied) on other commands.

***GET:-** A "get function" is available for some items, and can be used instead of the *GET command. The function returns the value and uses it where the function is input (bypassing the need for storing the value with a parameter name and inputting the parameter name where the value is to be used). For example, assume the average X-location of two nodes is to be calculated.

CM:- You can get ANSYS to write the components file by including a command CMWRITE,<jobname>.cm in your ANSYS input file, before the SOLVE command. Job names need to be consistent across restarts, input file processing and regular runs if .cm files are to be loaded into ANSYS CFX-Post.

NSLE :-selects NodeType nodes attached to the currently-selected set of elements. Only nodes on elements in the currently-selected element set can be selected.

EMAT:- File.EMAT is also required in these cases:

- For the static solution of a prestressed modal analysis when you use stress-stiffening energy to obtain better strain energy calculation accuracy.

- When you perform a QR damp eigensolution with a constant material damping coefficient

SEOPT:- Specifies substructure analysis options (ANTYPE,SUBSTR). If used in SOLUTION, this command is valid only within the first load step.

MXPAND:- Specifies the number of modes to expand and write over a frequency range for a modal (ANTYPE,MODAL) or buckling (ANTYPE,BUCKLE) analysis. For reduced analyses, an expansion is required. If used in SOLUTION, this command is valid only within the first load step.

There is no limit on the number of expanded modes (NMODE). However, there is a limit on the maximum number of modes used via the *GET,,MODE command, mode combinations, and the MDAMP command.

NUMEXP: Specifies solutions to be expanded from reduced analyses.

GROPT: Sets various line graph display options.

PLCAMP: To take the gyroscopic effect into account when plotting a Campbell diagram, first issue the CORIOLIS command in the SOLUTION module.

The PLCAMP command is valid only for modal analyses (QR damped [MODOPT,QRDAMP] or damped [MODOPT,DAMP] methods only). The command works over two or more load step results generated with an ascending order of rotational velocity (OMEGA or CMOMEGA).

Complex Eigen values are necessary (MODOPT,QRDAMP,,,Cpxmod = ON), and you must specify the number of modes to expand (MXPAND) at each load step.

In some cases where modes are not in the same order from one load step to the other, sorting (Option = 1) the frequencies can help to obtain a correct plot. Sorting is based on the comparison between complex mode shapes calculated at two successive load steps.

IX.2. Input Analysis in ANSYS

```
/batch, list
/title, Simple Rotor Supported by CMS Superelement
/filename, Rotor_Supp
! ** parameters
FL=0.79 ! foundation length
FW=0.154 ! foundation width
FT=0.025 ! foundation thickness
EX_F=70e9 ! foundation Young's modulus
dens_F=2710 ! foundation Density
FSH=0.25 ! support Height
FST=0.025 ! support Thickness
SCH=0.2 ! shaft Centre height
BOR=0.025 ! bearing OuterRadius =0.025 m
BS=5e7 ! bearing stiffness N/m
! ** disc 1
m1=0.334
J1=2.688e-4
I1=1.36e-4
! ** disc 2
m2=0.782
J2=7.02e-4
I2=4.31e-4
! ** disc 3
m3=4.39
```

```
Ix3=1.39e-2
Iy3=1.194e-2
Iz3=8.809e-3
EX_S=200e9
Dens_S=7850
S_OR=0.015

/prep7

! ** foundation
block, 0, FW, 0, FT, 0, FL
block, 0, FW, 0, FSH, (0.23-FST/2), (0.23+FST/2)
block, 0, FW, 0, FSH, (0.668-FST/2), (0.668+FST/2)
wpcsys, -1, 0
wpoff, FW/2, SCH
cswpla, 11, 1, 1, 1,
cyl4, , ,BOR, , ,FL
vsbv, 2, 4, , ,keep
vsbv, 3, 4, ,
vovlap,all
wpcsys, -1, 0
csys, 0
n,1,FW/2,0.2,0.073
n,2,FW/2,0.2,0.23
n,3,FW/2,0.2,0.23+0.147
n,4,FW/2,0.2,0.23+0.147+0.179
n,5,FW/2,0.2,0.23+0.147+0.179+0.112

! ** Shaft
et,1,188
keyopt,1,3,3
sectype,1,BEAM,CSOLID,Shaft, 0
secoffset,CENT
secdata,S_OR
ex,1,EX_S
nuxy,1,0.3
dens,1,Dens_S
r,1,
e,1,2
*repeat,4,1,1
et,2,21
r,2, m1,m1,m1,J1,I1,I1
r,3,m2,m2,m2,J3,I2,I2
r,4,m3,m3,m3,Ix3,Iy3,Iz3
type, 2 ! motor
real,4
e,1
real,2 ! disc 1
e,3
real,3 ! disc 2
e,4
et,5,185
keyopt,5,2,3
ex,5,EX_F
nuxy,5,0.3
dens,5,Dens_F
r,5
type,5
mat,5
real,5
csys,11
lsel,s,loc,x,0.025
lsize,all, , ,4
csys,0
lsel,s,loc,y,0.1*FT,0.9*FT
```



```

lesize,all, , ,2
lsel,s,loc,z, (0.23-FST/3), (0.23+FST/3)
lesize,all, , ,2, ,1
lsel,s,loc,z, (0.668-FST/3), (0.668+FST/3)
lesize,all, , ,2, ,1
allsel
esize,0.05
vsweep,all
*get, MAXNode, NODE,, NUM,MAX
!** bearings radial stiffness modeling
et,7,14
r,7,BS/4
type,7
real,7
csys,11
nset,s,loc,x,0,BOR
nset,r,loc,z,0.23
e,2,node (BOR, 0, 0.23)
e,2,node (BOR, 45, 0.23)
e,2,node (BOR, 90, 0.23)
e,2,node (BOR, 135, 0.23)
e,2,node (BOR, 180, 0.23)
e,2,node (BOR, 225, 0.23)
e,2,node (BOR, 270, 0.23)
e,2,node (BOR, 315, 0.23)
nset,s,loc,x,0,BOR
nset,r,loc,z,0.668
e,5,node (BOR,0,0.668)
e,5,node (BOR,45,0.668)
e,5,node (BOR,90,0.668)
e,5,node (BOR,135,0.668)
e,5,node (BOR,180,0.668)
e,5,node (BOR,225,0.668)
e,5,node (BOR,270,0.668)
e,5,node (BOR,315,0.668)
allsel

!**components
esel,s,ename, ,188
esel,a,ename, ,21
cm,ROTOR,elem ! rotating parts
esel,a,ename, ,14
nsle,s,1
cm,Rotor_Bear,Elem ! rotor+bearings (non SE)
esel, inve
cm,Support,Elem ! foundation (SE)
allsel
save,Rotor_Supp_Full, db
finish
!*****
! superelement Generation
!*****
/filename,Support
/solu
antype,substr
ematwrite,yes
seopt, Support, 1,
cmsopt, fix, 4
nset, s, loc,x, BOR
m,all,all
csys,0 ! fix bottom of the support structure
nset, s, loc, y, 0
d,all,all
cmsel,S,Support,Elem
nsle,s

solve !generate superelement "Support.sub"
fini
save
!*****
! Rotor+Bearings+Superelement modal analysis
!*****
/clear
/filename,Rotor_use
resume, Rotor_Supp_Full,db ! resume original
assembly model
/prep7
vclear,all ! delete foundation elements to be replaced by
superelement
et,10,matrix50
type,10
se,Support ! read in superelement from file
"Support.sub"
allsel
/solu
pi=acos (-1)
RPMtoRPS=2*pi/60
antype,modal
modopt,qrdamp,6,1,0,1,on
mxpand,6, , , 1
coriolis, on, , , on
beta, 1e-6
cmomega,rotor,0*RPMtoRPS, , ,
FW/2,SCH,0.073,FW/2,SCH,0.668
solve
cmomega,rotor,1000*RPMtoRPS, , ,
FW/2,SCH,0.073,FW/2,SCH,0.668
solve
cmomega,rotor,5000*RPMtoRPS, , ,
FW/2,SCH,0.073,FW/2,SCH,0.668
solve
cmomega,rotor,10000*RPMtoRPS, , ,
FW/2,SCH,0.073,FW/2,SCH,0.668
solve
save, Rotor_use, db
finish
!*****
! Expand results for Superelement
!*****
/clear
/filename, Support
resume, Support, db
/solu
antype,substr
expass,on
seexp, Support , Rotor_use,
numexp,all, , , yes
solve
finish
! *****
! Review the Results
! *****
/clear
!**combined results of foundation and rotor
/clear
/verify
resume, Rotor_use, db
/post1
/eshape,1
cmsfile,clear
file, Rotor_use, rst

```



```
set, 3, 2
plnsol,u,sum,2
cmsfile,add, Support.rst
set, last
esel,s,ename, ,185
plnsol,u,sum,2
fini
```

Modal analysis procedure

Modal analysis is a technique used to determine a structure's vibration characteristics such as natural frequencies, mode shapes, and mode participation factors. This is the most fundamental of all the dynamic analysis types.

Benefits of modal analysis

- Allows the design to avoid resonant vibrations or to vibrate at a specified frequency (speakers, for example).
- Gives engineers an idea of how the design will respond to different types of dynamic loads.
- Helps in calculating solution controls (time steps, etc.) for other dynamic analyses.
- Because a structure's vibration characteristics determine how it responds to any type of dynamic load, always perform a modal analysis first before trying any other dynamic analysis.
- Several mode extraction methods are available in ANSYS:
 - Block Lanczos (default)
 - Subspace
 - Power Dynamics
 - Reduced
 - Unsymmetric
 - Damped (full)
 - QR Damped

We are using the QR damped method among the above methods for modal analysis.

- A mode extraction method that considers damping effects is the QR damped method.
 - Faster and more stable than the existing Damped Solver
 - Works with poorly conditioned models
 - All forms of damping allowed including damper elements
 - Combines the best features of the real eigensolution method (Block Lanczos) and the Complex Heisenberg method (QR Algorithm)
 - Outputs complex eigenvalues (frequency and stability) and damping ratio of each mode
 - Supports the use of a material dependent damping ratio [MP, DMPR] in a subsequent mode superposition harmonic analysis.

Four main steps in a modal analysis:

- Build the model
- Choose analysis type and options
- Apply boundary conditions and solve
- Review results.
- These four steps are performed in the above modal analysis using command prompt in ANSYS.

Output screens

The analysis above will give the following image files as its result:

X. CONCLUSION

We have designed a rotor and performed modal analysis on it. We performed modal analysis using QR damped method and sub structuring by using command prompt in ANSYS

Thus we were able to predict the behavior of a rotor on axially symmetric linear rotating shaft between two bearings for the first mode shape of modal analysis.

Further scope of work: We will be performing rotor dynamic analysis on the same modal by considering damping, bearing stiffness and gyroscopic couple from which the critical speed of a rotor can be determined by drawing Campbell diagram.

REFERENCES

- [1] <http://mechanicalindia.blogspot.in/2009/02/latest-developments-in-mechanical.html>
- [2] <http://www.michigan.gov/careers/0,1607,7-170-46398-64748--,00.html>
- [3] http://www.kxcad.net/ansys/ANSYS/ansyshelp/ffcontact_tut.html
- [4] Rotor dynamics book written by J.S.Rao
- [5] We refer Investigating ANSYS Rotor dynamics by Jerome Montgomer(a paper published)
- [6] We refer Rotor dynamic analysis of 3D-modeled gas turbine by Joakim Samuelsson.
- [7] <http://www.scribd.com/doc/33750426/Rotor-Dynamics-With-ANSYS-Mechanical-Solutions>
- [8] <http://www.scribd.com/doc/37038268/ANSYS-V11-Rotor-Dynamics-Web-Seminar>
- [9] <https://confluence.cornell.edu/display/SIMULATION/ANSYS+11+-+Crank#ANSYS11-Crank-Step5%3AMeshgeometry>
- [10] <http://hochschule16.tk/>
- [11] www1.ansys.com/customer/content/documentation/120/ans_rot.pdf