Rotordynamisk analys av 3D-modellerad gasturbinrotor i ANSYS

Rotor dynamic analysis of 3D-modeled gas turbine rotor in ANSYS

Joakim Samuelsson

Solid Mechanics

Vårterminen 2009
Rotor dynamic analysis of 3D-modeled gas turbine rotor in Ansys

Joakim Samuelsson

Solid Mechanics

Department of Management and Engineering

Supervisor: Thomas Domeij
GRCRM, Siemens Industrial Turbomachinery AB

Examiner: Kjell Simonsson
IEI, Linköping University

Finspång, 8 juni, 2009
Preface

This report is the result of my master thesis work in Mechanical Engineering at Linköping’s University. The thesis work was done during the spring 2009 at Siemens Industrial Turbomachinery AB in Finspång.

During the thesis work I have got a lot of help and would like to thank them. On Siemens Industrial Turbomachinery AB, I want to thank Thomas Domeij at GRCRM for being a great mentor. I also want to thank Susanna Lundgren at GRCRM for all the help with Ansys and Pontus Welinder at GRCRM for helping me with Ardas. At Linköping’s University I would like to thank Kjell Simonsson at IEI for being my mentor and examiner.

This is an official version of the report and it is available for everybody on the Internet.

Finspång 2009-06-08

Joakim Samuelsson
Abstract

The world we are living in today is pushing the technology harder and harder. The products need to get better and today they also need to be friendlier to the environment. To get better products we need better analysis tools to optimize them and to get closer to the limit what the material can withstand. Siemens industrial Turbomachinery AB, at which thesis work is made, is constructing gas and steam turbines. Gas and steam turbines are important in producing power and electricity. Electricity is our most important invention we have and most of the people are just taking electricity for granted. One way to produce electricity is to use a gas turbine which is connected to a generator and by combing the turbine with a steam turbine the efficiency can be up to 60%. That is not good enough and everybody want to get better efficiency for the turbines, meaning less fuel consumption and less impact on the environment.

The purpose of this thesis work is to analyze a tool for rotor dynamics calculations. Rotor dynamics is important in designing a gas turbine rotor because bad dynamics can easily lead to disaster. Ansys Classic version 11 is the analyze program that is going to be evaluated for the rotor dynamic applications. Nowadays rotor dynamics is done with beam elements i.e. 1D models, but in this thesis work the beam elements are going to be changed to solid elements. With solid elements a 3D model can be built and thanks to that more complex calculations and simulations can be made. For example, with a 3D model 3D effects can be shown and e.g. simulations with blade loss can be done. 3D effects are not any problem today but in the future the gas turbines have to get better and maybe also the rotational speed will increase.

Ansys isn’t working perfectly yet, there are some problems. However Ansys have a good potential to be an additional tool for calculations of rotor dynamics, because more complex calculations and simulations can be done. More knowledge and time needs to form the rules to modeled a rotor and developing the analysis methods. Today the calculated lateral critical speeds are lower than the ones obtained from the in-house program Ardas version 2.9.3 which is used in Siemens Industrial Turbomachinery AB today. The difference between the programs are not so big for the four first lateral modes, only 3-8%, but the next three lateral modes have a difference of 10-20%. The torsion frequencies from Ansys are the same as the ones from Ardas, when the Solid186 elements are used to model the blades.
Sammanfattning


Ansys fungerar inte perfekt idag och programmet har några problem. Men Ansys har stor chans att bli ett hjälpprogram i rotordynamiska beräkningar för mer komplexa analyser och simuleringar. Det behövs mer tid och erfarenhet för att kunna forma reglerna hur man ska modellera olika rotorer och mer tid behövs för att utveckla de olika analysverktygen i Ansys. Idag är de lateralala moderna lägre än samma moder i Ardas version 2.9.3, som är beräkningsprogrammet som används idag på Siemens Industrial Turbomachinery AB. De fyra första lateral moderna har 3-8 % lägre kritisk hastighet medan de nästa tre har en skillnad på 10-20 % jämfört med Ardas. Torsionsmoderna i modellen från Ansys, som använder Solid186 elementet till skovlarna, har samma frekvens som Ardas.
# Table of contents

1 Introduction ........................................................................... 1
   1.1 The Company....................................................................................................... 1
   1.2 Aim and Purpose .............................................................................................. 1
   1.3 Limitations ........................................................................................................... 1
   1.4 Method ................................................................................................................. 1
   1.5 Gas turbine ........................................................................................................... 2
      1.5.1 Compressor ................................................................................................... 3
      1.5.2 Combustion chamber .................................................................................... 3
      1.5.3 Turbine .......................................................................................................... 3
      1.5.4 Blades ............................................................................................................ 3
      1.5.5 Stator ............................................................................................................. 3
      1.5.6 Bearings ........................................................................................................ 4

2 Rotor dynamics ...................................................................... 6
   2.1 Torsional Vibrations ............................................................................................ 6
   2.2 Longitudinal vibrations ........................................................................................ 7
   2.3 Lateral Vibrations ................................................................................................ 7
   2.4 Campbell diagram ................................................................................................ 7
   2.5 Gyroscopic stiffening effect ................................................................................. 8
   2.6 Whirling ............................................................................................................. 11
   2.7 Modes.................................................................................................................. 11
   2.8 Orbits.................................................................................................................. 13
   2.9 The gyroscopic effect and the mass effect ......................................................... 14
   2.10 Damping ........................................................................................................... 15
   2.11 Effects of the stiffness of the support .............................................................. 16
   2.12 Stability analysis .............................................................................................. 16

3 Ansys ..................................................................................... 17
   3.1 Stationary reference frame vs. rotating reference frame .................................... 17
      3.1.1 Stationary reference frame .......................................................................... 17
      3.1.2 Rotating reference frame ............................................................................. 17
   3.2 The equation of motion for rotating- and stationary reference frame ................. 18
      3.2.1 Rotating reference frame equation .............................................................. 20
      3.2.2 Stationary reference frame equation ........................................................... 21
   3.3 Units in Ansys .................................................................................................... 21
   3.4 Modal analysis in Ansys .................................................................................... 21
   3.5 Harmonic force response analysis in Ansys ...................................................... 23
      3.5.1 The Full method (Full) ................................................................................ 23
      3.5.2 Sparse solver ............................................................................................... 23
      3.5.3 The Mode Superposition method (MSUP) ................................................. 24
      3.5.4 Chosen method for the harmonic response analyses in Ansys ................. 24
   3.6 Damping in Ansys.............................................................................................. 24
   3.7 Mass unbalance forces ....................................................................................... 25

4 Verification of Ansys with simple models .......................... 26
   4.1 Case one ............................................................................................................. 26
   4.2 Case two: Verification between Ansys and Ardas with bearings ...................... 27
   4.3 Case three: Verification between Ansys and analytical model .......................... 28
      4.3.1 Analytical model for Case three ................................................................. 29
      4.3.2 Ansys model for Case three ........................................................................ 29
A.5.1 Batch file for the modal analysis for Case three ....................................... - 11 -
A.5.2 Batch file for the response analysis for Case three ............................... - 13 -
A.5.3 Batch file for the modal analysis for Case four ....................................... - 16 -
A.5.4 Batch file for the response analysis for Case four ............................... - 18 -
A.5.5 Batch file for the modal analysis for Case five ....................................... - 20 -
A.5.6 Batch file for the response analysis for Case five ............................... - 23 -
A.5.7 Matlab file for Case three....................................................................... - 25 -
A.5.8 Matlab file for Case four.......................................................................... - 27 -
A.5.9 Matlab file for Case five.......................................................................... - 29 -
# Table of figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Gas turbine</td>
<td>2</td>
</tr>
<tr>
<td>2</td>
<td>Journal bearing types</td>
<td>5</td>
</tr>
<tr>
<td>3</td>
<td>Campbell diagram</td>
<td>8</td>
</tr>
<tr>
<td>4</td>
<td>Angular momentum</td>
<td>9</td>
</tr>
<tr>
<td>5</td>
<td>Cylindrical modes</td>
<td>12</td>
</tr>
<tr>
<td>6</td>
<td>Conical modes</td>
<td>13</td>
</tr>
<tr>
<td>7</td>
<td>Orbit curve</td>
<td>14</td>
</tr>
<tr>
<td>8</td>
<td>Rotor with center disk and overhung disk</td>
<td>14</td>
</tr>
<tr>
<td>9</td>
<td>Coordinate system</td>
<td>18</td>
</tr>
<tr>
<td>10</td>
<td>Spin softening effect</td>
<td>20</td>
</tr>
<tr>
<td>11</td>
<td>Case one model</td>
<td>26</td>
</tr>
<tr>
<td>12</td>
<td>Combi214</td>
<td>27</td>
</tr>
<tr>
<td>13</td>
<td>Case three model</td>
<td>28</td>
</tr>
<tr>
<td>14</td>
<td>Mass center</td>
<td>28</td>
</tr>
<tr>
<td>15</td>
<td>Campbell diagram from a modal analysis for Case three</td>
<td>30</td>
</tr>
<tr>
<td>16</td>
<td>Response analysis for Case three</td>
<td>31</td>
</tr>
<tr>
<td>17</td>
<td>Campbell diagram from a modal analysis for Case four</td>
<td>33</td>
</tr>
<tr>
<td>18</td>
<td>Response analysis for Case four</td>
<td>34</td>
</tr>
<tr>
<td>19</td>
<td>Campbell diagram from a modal analysis for Case five</td>
<td>35</td>
</tr>
<tr>
<td>20</td>
<td>Response analysis for Case five</td>
<td>36</td>
</tr>
<tr>
<td>21</td>
<td>Ansys model of the gas turbine rotor</td>
<td>37</td>
</tr>
<tr>
<td>22</td>
<td>Stiffness curve for bearing 1</td>
<td>39</td>
</tr>
<tr>
<td>23</td>
<td>Stiffness curve for bearing 2</td>
<td>39</td>
</tr>
<tr>
<td>24</td>
<td>Damping curve for bearing 1</td>
<td>40</td>
</tr>
<tr>
<td>25</td>
<td>Damping curve for bearing 2</td>
<td>40</td>
</tr>
<tr>
<td>26</td>
<td>2D model of the rotor</td>
<td>42</td>
</tr>
<tr>
<td>27</td>
<td>First lateral mode at an undamped modal analysis</td>
<td>42</td>
</tr>
<tr>
<td>28</td>
<td>Second lateral mode at an undamped modal analysis</td>
<td>42</td>
</tr>
<tr>
<td>29</td>
<td>Third lateral mode at an undamped modal analysis</td>
<td>43</td>
</tr>
<tr>
<td>30</td>
<td>Mass unbalance</td>
<td>44</td>
</tr>
<tr>
<td>31</td>
<td>Showing the result of the retrograde and progressive motion</td>
<td>3</td>
</tr>
</tbody>
</table>
A list over the tables

Table 1: The influences of changed disk mass and radius for center disk rotor ..........15
Table 2: The influences of changed disk mass and radius for overhunged rotor ..........15
Table 3: Comparing stationary ref. frame with rotating ref. frame [11] ..................18
Table 4: Table over units in Ansys ........................................................................21
Table 5: Results Case one ..........................................................................................27
Table 6: Result Case two .............................................................................................27
Table 7: Ansys and Ardas are compared with each other .........................................49
Table 8: Comparing the different modal extracting tools in Ansys [11] .............. - 5 -
**Words and abbreviations**

[1] is the first reference in the reference list

Coriolis,on means that it is a commando in Ansys

SIT AB= Siemens Industrial Turbomachinery AB

DOF= Degree of freedom

### Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a_s$</td>
<td>acceleration in a stationary reference frame</td>
<td>[m/s$^2$]</td>
</tr>
<tr>
<td>$a_t$</td>
<td>acceleration in a rotating reference frame</td>
<td>[m/s$^2$]</td>
</tr>
<tr>
<td>$A$</td>
<td>major axis</td>
<td>[m]</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>mass matrix multiplier</td>
<td>[-]</td>
</tr>
<tr>
<td>$B$</td>
<td>minor axis</td>
<td>[m]</td>
</tr>
<tr>
<td>$\beta$</td>
<td>stiffness matrix multiplier</td>
<td>[-]</td>
</tr>
<tr>
<td>$\beta_k$</td>
<td>variable stiffness matrix multiplier</td>
<td>[-]</td>
</tr>
<tr>
<td>$\beta^s$</td>
<td>constant stiffness matrix coefficient</td>
<td>[-]</td>
</tr>
<tr>
<td>$\beta_j^m$</td>
<td>stiffness matrix multiplier for material j</td>
<td>[-]</td>
</tr>
<tr>
<td>$\beta_j^v$</td>
<td>constant stiffness matrix coefficient for material j</td>
<td>[-]</td>
</tr>
<tr>
<td>$c$</td>
<td>damping constant</td>
<td>[Nms/rad]</td>
</tr>
<tr>
<td>$c_\psi$</td>
<td>damping constant in $\psi$-direction</td>
<td>[Nms/rad]</td>
</tr>
<tr>
<td>$c_\theta$</td>
<td>damping constant in $\theta$-direction</td>
<td>[kg/s]</td>
</tr>
<tr>
<td>$[C]$</td>
<td>damping matrix</td>
<td>[kg/s]</td>
</tr>
<tr>
<td>$[C_k]$</td>
<td>element damping matrix</td>
<td>[kg/s]</td>
</tr>
<tr>
<td>$[C_{\xi\xi}]$</td>
<td>frequency-dependent damping matrix</td>
<td>[kg/s]</td>
</tr>
<tr>
<td>$[C_{\xi\psi}]$</td>
<td>Gyroscopic matrix</td>
<td>[kg/s]</td>
</tr>
<tr>
<td>$[C_{\xi\phi}]$</td>
<td>frequency-dependent damping matrix</td>
<td>[kg/s]</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>the angle between axis of rotation and the disk axis</td>
<td>[m]</td>
</tr>
<tr>
<td>$d$</td>
<td>disk diameter</td>
<td>[m]</td>
</tr>
<tr>
<td>$D$</td>
<td>disk diameter</td>
<td>[m]</td>
</tr>
<tr>
<td>$\delta$</td>
<td>log decrement</td>
<td>[-]</td>
</tr>
<tr>
<td>$e$</td>
<td>distance between mass unbalance and axis of rotation</td>
<td>[m]</td>
</tr>
<tr>
<td>$E$</td>
<td>Young’s module</td>
<td>[Nm]</td>
</tr>
<tr>
<td>$f$</td>
<td>frequency</td>
<td>[Hz]</td>
</tr>
<tr>
<td>${F}$</td>
<td>force vector</td>
<td>[N]</td>
</tr>
<tr>
<td>$h$</td>
<td>disk thickness</td>
<td>[m]</td>
</tr>
<tr>
<td>${H}$</td>
<td>angular moment of inertia vector</td>
<td>[kgm$^2$/s]</td>
</tr>
<tr>
<td>$H_y$</td>
<td>angular moment of inertia in $y$-direction</td>
<td>[kgm$^2$/s]</td>
</tr>
<tr>
<td>$H_z$</td>
<td>angular moment of inertia in $z$-direction</td>
<td>[kgm$^2$/s]</td>
</tr>
<tr>
<td>$\theta$</td>
<td>skew angle of the disk in vertical direction</td>
<td>[radians]</td>
</tr>
<tr>
<td>$\theta_p$</td>
<td>angle for progressive motion</td>
<td>[radians]</td>
</tr>
<tr>
<td>$\theta_r$</td>
<td>angle for retrograde motion</td>
<td>[radians]</td>
</tr>
<tr>
<td>$I$</td>
<td>second moment of area</td>
<td>[m$^4$]</td>
</tr>
<tr>
<td>$J_p$</td>
<td>polar mass of inertia</td>
<td>[kgm$^2$]</td>
</tr>
<tr>
<td>$J_T$</td>
<td>transversal moment of inertia</td>
<td>[kgm$^2$]</td>
</tr>
<tr>
<td>$k$</td>
<td>stiffness constant</td>
<td>[N/m]</td>
</tr>
<tr>
<td>$k_\psi$</td>
<td>stiffness constant in $\psi$-direction</td>
<td>[Nm/rad]</td>
</tr>
<tr>
<td>$k_\theta$</td>
<td>stiffness constant in $\theta$-direction</td>
<td>[Nm/rad]</td>
</tr>
<tr>
<td>$[K]$</td>
<td>Stiffness matrix</td>
<td>[N/m]</td>
</tr>
</tbody>
</table>
[K_{spin}] spin softening matrix [kg/s]
L length [m]
m mass [kg]
m_u mass of mass unbalance [kg]
[M] mass matrix [kg]
N_e number of element with damping [-]
N_m number of materials [-]
\(\xi\) constant damping ratio [-]
\(\xi_d\) damping ratio [-]
\(\xi_m\) modal damping ratio [-]
p real part of complex eigenvalue [Hz]
r radius [m]
\{r\} distance in a stationary reference frame [m]
\{r^*\} distance in a rotating reference frame [m]
\{R\} distance between stationary-rotating reference frame [m]
\(\rho\) density [kg/m^3]
s complex eigenvalue [Hz]
t time [s]
T kinetic energy [Nm]
u_{max} maximum displacement [m]
\{u\} displacement vector [m]
v velocity [m/s]
v_r velocity in a rotating reference frame [m/s]
v_s velocity in a stationary reference frame [m/s]
V potential energy [Nm]
\(\nu\) poisson’s number [-]
\(\phi\) angle between \(\gamma\)-direction and \(\theta\)-direction [radians]
\{\omega\} rotational speed vector [radians/s]
\(\omega_i\) eigenfrequency for mode i [radians/s]
\(\omega_d\) imaginary part of complex eigenvalue [Hz]
\Phi\) displacement phase shift [radians]
\{\Phi\} mode shape [-]
x_p radius for progressive motion [m]
x_r radius for retrograde motion [m]
\(\Psi\) skew angle of the disk in horizontal direction [radians]
\(\Omega\) rotational speed [radians/s]
y y-coordinate [m]
z z-coordinate [m]
1 Introduction

This work is done on SIT AB in Finspång on the GRCRM department, which is working with the compressor part in gas turbines. On the department they are calculating rotor dynamics and that is what this thesis work is about. To improve the gas turbines today they need better analysis program to be able to do more complex calculations and simulations, and therefore Ansys Classic version 11 will be evaluated as an analysis program for rotor dynamics calculations.

In this section Siemens Industrial Turbomachinery AB is presented and also a gas turbine will be explained. The aim, limitations and method of the work are presented.

1.1 The Company

This thesis work was done at Siemens Industrial Turbomachinery in Finspång. Finspång is located about 60 km north of Linköping and 30 km west of Norrköping. SIT AB is producing gas and steam turbines for the whole world and SIT AB is part of Siemens which has over 427 000 employers in 190 countries. Siemens bought SIT AB from Alstom in 2005 and then was also the name changed to Siemens Industrial Turbomachinery AB. SIT AB has over 2400 employers in Finspång, 120 employers in Trollhättan and a turnover of about 7 billion SEK. SIT AB is producing four different gas turbines with an output power of 17 to 47 MW. SIT AB is also producing steam turbines which have an output power of 60 to 180 MW [1].

1.2 Aim and Purpose

The aim with this thesis work is to evaluate if Ansys Classic version 11 is a proper tool to use in rotor dynamics calculations. How the rotor should be designed in the software to get the best result should also be evaluated. The work is divided into three major parts. The first is a theory part about gas turbines, rotor dynamics and Ansys. The second part is to verify the program with simple models which are compared with theoretical calculations. The third major part is to model one of SIT AB’s gas turbine rotors in Ansys and evaluate how good the results are compared to other programs.

1.3 Limitations

Some limitations have to be done since otherwise the work would be too massive, therefore only the rotor will be modeled in Ansys, the stator will be ignored. Transient analysis is not going to be considered either.

1.4 Method

The first part is to verify Ansys with different simple models where the results are compared to theoretical analyses. This is done to see that Ansys is working and to see if the results are reliable. The second part is to model a gas turbine rotor and to evaluate the result from Ansys and compare it with the result from Ardas version 2.9.3, which is the in-house program that is used today at SIT AB. Ardas is made by SIT AB and has been used for over 20 years. The program has been compared many times with different programs and measured values [2]. Different models of the rotor should be tested in Ansys to see how the model should be designed to get the best result. The results are then going to be analyzed and a recommendation will be given about how the program is working and if the results are satisfying.
1.5 Gas turbine

A gas turbine is a power engine with a large rotor driven by hot gas flowing pass the turbine. It is the same function as a jet engine on an airplane. Gas turbines can be divided into three different kinds of turbines, which depend on how many shafts the turbine has. A turbine can have one, two or three shafts. How many shafts a turbine has is important in rotor dynamics because the shafts can generally be calculated independently from each other. Gas turbines are popular thanks to high power/weight ratio compared to other engines like diesel engines. Turbines have few moving parts which means they become more reliable. Maintenance cost is low and it is cheaper and easier to install a power plant with gas turbines than other power plants, like ones that are using coal and nuclear material. This the reason why gas turbines are popular in developing countries which need electrical power fast. To install a basic gas turbine power plant takes only 10-12 months. This can be compared with 48-60 months for a nuclear power plant and 36-42 months for a coal steam power plant. Another advantage with gas turbines is that they can start and stop in only a few minutes. Gas turbines are mostly used for producing power to oil pipes and electricity by generators, but they are also used in helicopters, boats and aircrafts. The fuels are natural gas, distillate oil, crude oil or residual oil [3]. The highest efficiency number today for a gas turbine is 46 % and SIT’s gas turbines have 37 % [4]. By combining a gas turbine with a steam turbine the efficiency can be up to 60 % and if also the warm water is taking care of, for example by warming up houses, the efficiency can be up to 85 %. To increase the efficiency more, the pressure ratio in the compressor needs to be higher and the inlet air temperature to the turbine blades has to be higher. The problem with higher air temperature is the materials, in the turbine blades, which may not withstand too extreme temperatures. Thus, the materials have to be improved and the cooling system has to get better to increase the efficiency. A picture of a gas turbine is shown in Figure 1 [3].
1.5.1 Compressor
The compressor is the first part of the turbine rotor after the air inlet. SIT’s gas turbines have an axial flow compressor. The idea with axial flow compressor is to first accelerate the air and then diffuse it into high pressure. The air accelerates by the blades on the rotor and is then diffused by a row of stationary blades which are located on the stator wall. During the diffusion the high velocity is changed into high pressure. One row of blades on the rotor and one row of blades on the stator is a stage, and every stage can compress the air 1.1-1.4 times. To obtain a pressure ratio of 20 the compressor needs to have many stages. A really good compressor can increase the pressure 40 times over all the stages. When the air gets compressed it takes less room and that is the reason why the rotor is conically shaped over the compressor part. It is important that the efficiency of the compressor is high because the compressor is driven by the turbine and therefore taking power from it [3].

1.5.2 Combustion chamber
When the compressed air leaves the compressor it enters the combustion chamber where the air is diffused to lower velocity. The velocity has to be low for avoiding the air to take the flame downstream. The velocity of the air leaving the compressor is about 122-183 m/s and the velocity of the air can not be higher than 15 m/s when it’s passing the flame. The flame is only heating up a part of the total air volume and then the heated air is mixed with the rest of the air. After the air has been mixed together it flows against the turbine blades with a temperature up to 1600 °C and with a high velocity because the air is accelerated by a nozzle [3].

1.5.3 Turbine
This is at the end of the rotor and it is here the power is taken out when the air is flowing pass the blades. The temperature of the air is high and therefore the blades have to be made of special material and with a cooling system. Not all the power from the turbine blades can be used by an external machine because the turbine blades also have to power the compressor [3].

1.5.4 Blades
The blades are the fan blades that are placed on the compressor and on the turbine disks. The blades are the blades which are transporting the air in the compressor into to the combustion chamber. The blades, which are located on the compressor, are made in such a way that the velocity of the air increases. The blades that are located on the turbine disk are specially made so they can withstand the hot air passing by [3].

1.5.5 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). Because of the hug power from the turbine, the stator part also needs to be added in the calculation of rotor dynamics because the stator can not be approximated as fixed [3].
1.5.6 Bearings

The bearings are supporting and holding the rotor in the right position in the stator. SIT AB is using journal bearings which are fed by oil to hold the rotor in place [2]. 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 [3].

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

In heavy machine like gas turbines, journal bearings are used. The bearing can be of different shape. In Figure 2 different bearing type are shown.

- The plain cylindrical has just a cylindrical shape and is cheap to produce but it has bad rotor dynamics properties.

- The Partial arc is a version of the plain cylindrical where the lower side of the bearing is cut away. This has somewhat better rotor dynamics properties than the plain.

- The two-axial-groove is also a version of the plain but it has two small cuts on both sides of the cylinder. This improves the stability and the rotor dynamics properties.

- The elliptical is very similar to the two-axial-groove except it has an elliptical form which gives better stability properties.

- There is also the three-lobe which has better rotor dynamics properties than the two-axial-groove.

- Offset cylindrical consist of two cylinders which are moved offset from each other and that makes the stability properties very good, but the rotor can only rotate in one direction.

- 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. SIT AB is using Tilting Padding for all their gas turbines [3].
Figure 2: Journal bearing types
2 Rotor dynamics

Many years ago engines didn’t operate with any high velocity and because of that engines had less stability problems. Today the turbines have to be more efficient, that means that they need to have higher rotational speed and because of that the turbines may get stability problems. Stability problems are the reason why rotor dynamics are important when developing gas turbines. Any stability problem in the rotor can easily lead to disaster and end up to be very expensive. The engineer wants to avoid oscillations in a system because oscillations can shorten the lifetime of the machine. Oscillations can also make the environment around the machine intolerable with heavy vibrations and high sound [3].

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 [5]. 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 can not always be calculated separately from each other because sometimes they are affecting each other [2][3].

2.1 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. If there is an external torque acting on the rotor, which frequency is the same as one of the torsional modes, torsional fatigue can happen with crack progression [2][5].
2.2 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 [5].

2.3 Lateral Vibrations

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 eigenvalues. 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, section 5.4. Complex eigenvalues analyses are done to get the critical speeds and that is done by setting up a Campbell diagram, see section 2.4. The analysis is also called damped critical speeds, section 5.4. 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. When an engineer is dealing with lateral vibrations the stator has to be in the model too, because it can not be approximated as fixed. The stator is affecting the rotor dynamics of the rotor during operation. Stability analysis is done by looking at the real parts of complex eigenvalues from a complex eigenvalues analysis. If the real part is lower than zero the rotor is stable and if it is larger than zero it is unstable [2][5].

2.4 Campbell diagram

The most important tool an engineer has when critical speeds should be decided is a Campbell diagram [6]. The diagram has the rotational speed of the rotor on the x-axis and the mode frequencies on the y-axis [3]. The modes, see section 2.7, 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, see section 2.6, modes are increasing and backward whirling modes are decreasing which is shown in the diagram in Figure 3. 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. An extra line is plotted in Figure 3 and it is the excitation line which represents natural excitations acting on the rotor. It can be an external forces or mass unbalance in the rotor. The excitation line in Figure 3 is synchronized with the rotational speed. The critical speeds are where the excitation line crosses any of the mode lines. Therefore in the diagram, in Figure 3, there are four critical speeds. The torsional mode can also been seen as a green straight line at the top of the diagram in Figure 3 [3].
Campbell diagram is from Ansys Classic v11. The blue straight line is the excitation line from the rotor itself. FW is forward whirling and BW is backward whirling

**2.5 Gyroscopic stiffening effect**

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 effect is 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, see section 2.6, 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 [7]. Where the gyroscopic effects are coming from will be explained by a simple model. The rotational speed of the disk is in the x-direction with the value \( \Omega \) and the skew angles for the disk in the y-and z-directions are \( \psi \) and \( \theta \) respectively. \( J_p \) is the polar mass moment of inertia in the direction of axis of rotation, \( J_T \) is the transversal moment of inertia and \( H \) is the angular momentum. The arrows, in the Figure 4, are representing angular velocity and angular momentum except for \( \psi \) and \( \theta \) because they are only representing angles.
The kinetic energy, Equation 4, for transversal rotation of the disk is set up and with that equation the gyroscopic terms can be explained with some steps. The disk center is located in the center of the coordinate system. The disk has rotation in the y- and z-direction. The skew angles are consider small, and therefore are \( \cos \theta = 1 \) and \( \sin \theta = 0 \), same thing with \( \psi \). Equation 1 is the general kinetic equation for a rigid body motion. The equations from 2 to 3 are coming from [7] and equations from 4 to 13 are coming from [8].

\[
T = \frac{1}{2} m v^2 + \frac{1}{2} \{\omega\} \cdot \{H\} 
\]

Where:
- \( T \) = kinetic energy
- \( m \) = mass of rotor
- \( v \) = transversal velocity
- \( \{\omega\} \) = column matrix over the rotational speed of the disk
- \( \{H\} \) = angular momentum of disk

The transversal velocities are zero and therefore the kinetic energy for transversal rotation is only interesting. The angular momentum equations of the disk are obtained in Figure 4 and written in Equation 2.

\[
\{H\} = \begin{bmatrix} H_y \\ H_z \end{bmatrix} = \begin{bmatrix} -J_T \psi + J_p \Omega \theta \\ J_T \dot{\theta} + J_p \Omega \psi \end{bmatrix} 
\]

\[
\{\omega\} = \begin{bmatrix} \dot{\psi} \\ \dot{\theta} \end{bmatrix} 
\]

Using Equation 1 together with equations 2 and 3, the kinetic energy is defined for transversal rotation of the disk.
\[
T = \frac{1}{2} J_T \left( \ddot{\theta}^2 + \dot{\psi}^2 \right) - \frac{1}{2} J_p \Omega \left( \dot{\psi} - \dot{\theta} \right) \tag{4}
\]

Using Lagrange Equation 5 and 6 [10]. “\( V \)” is the potential energy and that is ignored because the disk isn’t moving anything in transversal direction.

\[
\frac{d}{dt} \left( \frac{dT}{d\theta} \right) - \frac{dT}{d\theta} + \frac{dV}{d\theta} = 0 \tag{5}
\]

\[
\frac{d}{dt} \left( \frac{dT}{d\psi} \right) - \frac{dT}{d\psi} + \frac{dV}{d\psi} = 0 \tag{6}
\]

\[
\frac{dT}{d\theta} = -\frac{1}{2} J_p \Omega \dot{\psi} \tag{7}
\]

\[
\frac{dT}{d\psi} = -\frac{1}{2} J_p \Omega \dot{\theta} \tag{8}
\]

\[
\frac{d}{dt} \left( \frac{dT}{d\theta} \right) = J_T \ddot{\theta} + \frac{1}{2} J_p \Omega \dot{\psi} \tag{9}
\]

\[
\frac{d}{dt} \left( \frac{dT}{d\psi} \right) = J_T \ddot{\psi} - \frac{1}{2} J_p \Omega \dot{\theta} \tag{10}
\]

Set equations 5 to 10 together to the Equations of angular motion which are shown in equations 11 and 12.

\[
\begin{align*}
J_T \ddot{\theta} + J_p \Omega \dot{\psi} &= 0 \tag{11} \\
J_T \ddot{\psi} - J_p \Omega \dot{\theta} &= 0 \tag{12}
\end{align*}
\]

The gyroscopic terms are \( J_p \Omega \dot{\psi} \) and \( -J_p \Omega \dot{\theta} \). The gyroscopic terms are dependent on the rotational speed and that is what make these terms important in rotating structures. Damping and stiffness can also be added to the equations 11 and 12, by which the Equations of angular motion, Equation 13, of the disk take the form.

\[
\begin{bmatrix}
J_T & 0 \\
0 & J_T
\end{bmatrix}
\begin{bmatrix}
\ddot{\theta} \\
\ddot{\psi}
\end{bmatrix}
+ \begin{bmatrix}
c_{\theta} & \Omega J_p \\
-\Omega J_p & c_{\psi}
\end{bmatrix}
\begin{bmatrix}
\dot{\theta} \\
\dot{\psi}
\end{bmatrix}
+ \begin{bmatrix}
k_{\theta} & 0 \\
0 & k_{\psi}
\end{bmatrix}
\begin{bmatrix}
\theta \\
\psi
\end{bmatrix}
= 0 \tag{13}
\]

Where k and c are here the stiffness and damping constant for transversal rotation of the disk.
2.6 Whirling

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 [2]. Backward whirling is when the whirling rotation is opposite the rotational speed. Which type of whirling motion the rotor has can also been seen in a Campbell diagram, see Figure 3, 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 [5].

2.7 Modes

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 gas turbine because the operating velocity can not 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. The lateral modes can also be divided into different groups which depend on how they look.

Cylindrical modes are the first modes and they can also be called rigid modes because the rotor doesn’t bend except for very stiff bearings. The cylindrical modes are shown in Figure 5. From a front view it looks like the rotor is only bouncing up and down. The natural frequency of a cylindrical mode is not changing so much with the rotational speed. The frequency on a non-rotating rotor is almost the same as one which is rotating with a high velocity. If a Campbell diagram is set up, see Figure 15 on the two first lateral mode lines, it can be shown that forward whirling and backward whirling are the same and the lines are not separating so much from each other. There is a minor gyroscopic effect on the disk. Sometimes can the whirling properties be seen when natural frequencies of the back whirling modes slightly are decreasing at higher rotational speed when the bearing stiffness is high.
The second type of lateral modes is conical modes and they are shown in Figure 6. The modes look like somebody is holding the center of the rotor still and the ends are moving in circles. The modes are sometimes called rocking modes or pitching modes. The differences on the natural frequencies between a non-rotating rotor and a rotor with high rotational speed are very big when the rotor has a conical motion. The whirling properties are playing an important role here when the backward whirling mode’s frequency decrease while forward mode’s frequency increase. This is because of the gyroscopic stiffening effect on the rotor when it is rocking. Consider forward whirling first; when the shaft rotational speed is increasing the gyroscopic stiffening effect is producing a moment that is affecting the disk and therefore becomes the rotor stiffer. The gyroscopic stiffening effect is only producing a moment on the disk when disk, which makes the rotor stiffer, has transversal rotation and the natural frequencies will increase. For backward whirling the gyroscopic effect is opposite, when the shaft’s rotational speed increases the gyroscopic stiffening effect is decreasing the stiffness of the rotor and that means that the natural frequencies decreases with increasing rotational speed. This phenomenon can be shown with a Campbell diagram, (see Figure 15 on the two highest lateral mode lines), where the forward modes and the backward modes are separated from each other [9].
2.8 Orbits
An orbit, shown in Figure 7, is the curve the rotor is moving in y- and z-direction when the axis of rotation is in the x-direction. If the bearing stiffness is set to the same value in horizontal and in vertical direction the orbit motion will be circular. If the stiffness is not the same in horizontal and vertical direction, the motion will be an elliptical curve instead of a circle. The elliptical curve has one major axial “A” and one minor axial “B”. The major axial is always positive and the minor axial is negative if it is a retrograde motion (backward whirling), this used in section 4.4-4.5 when response curves are drawn [7]. Why the gyroscopic stiffening effects are separating the forward and back whirling lines in a Campbell diagram is explained in appendix A.1.
2.9 The gyroscopic effect and the mass effect

The gyroscopic effect influences the rotor in such a way that the natural frequencies are changing with the rotational speed, but it depends also on which kind of mode motion the rotor has. To get an influence of the gyroscopic effect the rotor needs to have some conical motion. To show how the different type of modes are affected by changed radius and mass of the disk six example has been set up, a set of three of a center disk rotor and a set of three of a rotor with overhung disk.

![Figure 7: Orbit curve](image)

![Figure 8: Rotor with center disk and overhung disk](image)

Starting with a rotor with center disk, (the left rotor in Figure 8), where one is nominal and two are modified. The two modified ones will be compared to the nominal one. The first modified model has the same radius on the disk but the twice the mass as the nominal model. The second modified model has the same mass as the nominal model but the radius on the disk is reduced. The first mode is a cylindrical and the second mode is a conical. The influences of changed mass and radius of the disk on a center disk rotor are shown Table 1.
Table 1: The influences of changed disk mass and radius for center disk rotor

<table>
<thead>
<tr>
<th>Type modification on the disk on the center disk rotor</th>
<th>Type of mode</th>
<th>Natural frequency</th>
<th>Gyroscopic effect</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nominal</td>
<td>Cylindrical</td>
<td>Normal</td>
<td>Normal</td>
</tr>
<tr>
<td>Nominal</td>
<td>Conical</td>
<td>Normal</td>
<td>Normal</td>
</tr>
<tr>
<td>Higher disk mass, unchanged radius</td>
<td>Cylindrical</td>
<td>Lower</td>
<td>Normal</td>
</tr>
<tr>
<td>Higher disk mass, unchanged radius</td>
<td>Conical</td>
<td>Normal</td>
<td>Normal</td>
</tr>
<tr>
<td>Reduced radius of the disk, unchanged mass</td>
<td>Cylindrical</td>
<td>Normal</td>
<td>Normal</td>
</tr>
<tr>
<td>Reduced radius of the disk, unchanged mass</td>
<td>Conical</td>
<td>Increased</td>
<td>Lower</td>
</tr>
</tbody>
</table>

The second rotor with overhung disk, (the right rotor in Figure 8), is modified with two different properties which are compared with a nominal rotor. The disk is modified in the same way as in the center disk model. Both modes are here conical, and therefore they are both affected by the gyroscopic effect. The influences of changed mass and radius of the disk on an overhunged disk rotor are shown Table 2.

Table 2: The influences of changed disk mass and radius for overhunged rotor

<table>
<thead>
<tr>
<th>Type modification on the disk on the overhunged rotor</th>
<th>Type of mode</th>
<th>Natural frequency</th>
<th>Gyroscopic effect</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nominal</td>
<td>Conical</td>
<td>Normal</td>
<td>Normal</td>
</tr>
<tr>
<td>Nominal</td>
<td>Conical</td>
<td>Normal</td>
<td>Normal</td>
</tr>
<tr>
<td>Higher disk mass, unchanged radius</td>
<td>Conical</td>
<td>Lower</td>
<td>Normal</td>
</tr>
<tr>
<td>Higher disk mass, unchanged radius</td>
<td>Conical</td>
<td>Lower</td>
<td>Normal</td>
</tr>
<tr>
<td>Reduced radius of the disk, unchanged mass</td>
<td>Conical</td>
<td>Increased</td>
<td>Lower</td>
</tr>
<tr>
<td>Reduced radius of the disk, unchanged mass</td>
<td>Conical</td>
<td>Increased</td>
<td>Lower</td>
</tr>
</tbody>
</table>

To summarize how the changed radius and mass of the disk are affecting the cylindrical modes and the conical modes. The mass has the biggest influence on cylindrical modes, while changed radius has the biggest influence on conical modes where the disk is rocking. The effects of the gyroscopic moment are bigger with larger disk [9].

2.10 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 overdamped, underdamped or critically damped. When a system is underdamped it will stop in an oscillated motion and if it is overdamped it will stop without any oscillations. Critical damped is the lowest damping a system can have without any oscillations [3][10]. To know if the system is critically damped or
overdamped is important when a modal analysis should be done in Ansys. The system
can not be critically damped or overdamped with the QR Damped method, see section
3.4, and that is the method which is going to be used for the modal analyses [11].

2.11 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 decreases the whirl
amplitude at the critical speed [5].

2.12 Stability analysis
The solution of a damped modal analysis in Ansys is a complex value $s = p \pm iw_d$,
where the imaginary part is the frequency of the oscillation. The real part is the
damping exponent and the rotor is stable if it is negative. This value is not the best
way to compare damping with different modes and other machines. Another
measurement, called the log decrement ($\delta$), 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. Equation 14 is the log decrement equation
[12].

$$\delta = \frac{-2\pi p}{|\omega_d|}$$

(14)
3 Ansys

Ansys is a computer-aided engineering (CAE) program. During the thesis work only Ansys Classic version 11 has been used. Ansys has another version which is called Ansys Workbench which is a more modern work environment. However, Workbench is still missing some of the rotor dynamics functions that Classic has. The program is based on the Finite Element Method (FEM) and has been on the market for 35 years. Ansys Classic can be used with two different techniques, one is to use the menu in the program and another is to use batch files which are text files with commands. The batch technique has been used during this thesis work, since a server can be used for calculations [13].

3.1 Stationary reference frame vs. rotating reference frame

To analyze rotating models two different coordinate system can be used in Ansys, stationary and rotating. To choose the coordinate system is the first an engineer has to do when a rotating system should be analyzed because the coordinate system decides which kind of analyses can be done. The differences between a stationary reference frame and a rotating reference frame are shown in Table 3.

3.1.1 Stationary reference frame

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 gas 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, see section 3.4, 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 eigenfrequencies [11].

3.1.2 Rotating reference frame

When using a rotating reference frame everything in the model is rotating with the same rotational speed. The coordinate system is rotating together with the model so everything around the model is moving instead. The rotating reference frame is activated by command coriolis,on. The analyses which can be done are static, modal,
transient and harmonic force response. Static analyses are calculating the stresses and the displacements. A rotating reference frame is used for flexible body dynamics where there are no stationary parts. The Coriolis matrix and the spin softening matrix correspond to the gyroscopic matrix in a stationary reference frame. Calculating the Campbell diagram with a rotating reference frame is not recommended [11].

Table 3: Comparing stationary ref. frame with rotating ref. frame [11]

<table>
<thead>
<tr>
<th>Stationary reference frame</th>
<th>Rotating reference frame</th>
</tr>
</thead>
<tbody>
<tr>
<td>No static analysis</td>
<td>Can do static analysis</td>
</tr>
<tr>
<td>Campbell diagram can be used to get critical speeds</td>
<td>Campbell can not be used</td>
</tr>
<tr>
<td>The rotation parts have to be axis-symmetric</td>
<td>The model don’t have to be axis-symmetric</td>
</tr>
<tr>
<td>One part can rotate and another can be stationary</td>
<td>Everything in the model is rotating</td>
</tr>
<tr>
<td>Up to 100 different components can rotate with different rotational speeds</td>
<td>Only one model can rotate</td>
</tr>
<tr>
<td>Ansys is recommending to do modal analysis in a stationary reference frame</td>
<td></td>
</tr>
</tbody>
</table>

3.2 The equation of motion for rotating- and stationary reference frame

Figure 9: Coordinate system
A rotating body around the global coordinate system
The picture is based on [11]
In Figure 9 a body is rotating around the global coordinate system (x, y, z) with angular velocity \( \{\omega\} \) which is defined in a stationary reference frame. The position of the point P in the global coordinate system is \( \{r\} \) and the position in the rotating coordinate system (x*, y*, z*) is \( \{r^*\} \) [11]. The position of the point P with respect on a stationary reference frame is shown in equation 15.

\[
\{r\} = \{R\} + \{r^*\}
\]  

(15)

The velocity of the point P is found by differentiating Equation 15 with respect to time.

\[
\left\{ \frac{dr}{dt} \right\}_s = \{v_r\} \quad \text{The velocity of P in a stationary reference frame}
\]

\[
\left\{ \frac{dr^*}{dt} \right\}_r = \{v_r\} + \{\omega\} \times \{r^*\} \quad \text{The velocity of P in a rotating reference frame}
\]

\[
\left\{ \frac{dR}{dt} \right\}_s = \{V\} \quad \text{The velocity of the rotating coordinate system in the global coordinate system}
\]

\[
\{v_s\} = \left\{ \frac{dR}{dt} \right\}_s + \left\{ \frac{dr^*}{dt} \right\}_r = \{V\} + \{v_r\} + \{\omega\} \times \{r^*\}
\]

(16)

The acceleration of the point P is found by differentiating Equation 16 with respect to time.

\[
\{a_s\} = \{a\} + \{a_r\} + \left\{ \frac{\cdot \omega}{dt} \right\} \times \{r^*\} + \{\omega\} \times (\{\omega\} \times \{r^*\}) + 2\{\omega\} \times \{v_r\}
\]

(17)

Assume that centre of the rotating coordinate system is fixed in the body, then \( \{a\} = 0 \).

\[
\{a_s\} = \{a_r\} + \left\{ \frac{\cdot \omega}{dt} \right\} \times \{r^*\} + \{\omega\} \times (\{\omega\} \times \{r^*\}) + 2\{\omega\} \times \{v_r\}
\]

(18)

If also the angular velocity \( \omega \) is constant.

\[
\{a_s\} = \{a_r\} + \{\omega\} \times (\{\omega\} \times \{r^*\}) + 2\{\omega\} \times \{v_r\}
\]

(19)

If small displacements \( u \) are considered.

\[
\{r\} = \{r_0\} + \{u\}
\]

\[
\{r^*\} = \{r^*_0\} + \{u^*\}
\]

(20)

(21)

Then Equation 19 can be rewritten as
\[
\begin{align*}
\{\dddot{u}\} &= \{\dddot{u}\}^* + \{\omega\} \times (\{\omega\} \times \{r^*\}) + \{\omega\} \times (\{\omega\} \times \{u^*\}) + 2\{\omega\} \times \{u^*\} \\
&= \{\dddot{u}\}^* + \{\omega\} \times (\{\omega\} \times \{r^*\}) + \{\omega\} \times (\{\omega\} \times \{u^*\}) + 2\{\omega\} \times \{u^*\}
\end{align*}
\] (22)

2\{\omega\} \times \{u^*\} is the Coriolis acceleration. The Coriolis acceleration is perpendicular to the axis of rotation and the displacement velocity. Coriolis is only used in a rotating reference frame.

\{\omega\} \times (\{\omega\} \times \{u^*\}) is the spin softening effect shown in Figure 10. When a body is rotating around an axis the material is moving outwards because of the centrifugal force. The stiffness matrix will change when body parts are moving outwards. Spin softening is only allowed to use in a rotating reference frame. The new stiffness matrix is declared by Equation 23 and Equation 24.

\[
\begin{align*}
\{u\} \{K\}^{-1} &= \Omega^2 \{M\} \{r\} + \{u\} \\
\{u\} \{K\}^{-1} - \Omega^2 \{M\} &= \Omega^2 \{M\} \{u\}
\end{align*}
\] (23) (24)

The new stiffness matrix is \([K'] = [K] - \Omega^2 [M]

\{\omega\} \times (\{\omega\} \times \{r^*\}) is the centrifugal acceleration. Centrifugal acceleration is pointing outwards from the axis of rotation and tries to move parts in the rotor outwards [11].

**3.2.1 Rotating reference frame equation**

\[
\begin{align*}
\{M\} \{\dddot{u}\} + \{C\} + \{C_{cor}\} \{\dddot{u}\} + \{K\} - \{K_{spin}\} \{u^*\} &= \{F\}
\end{align*}
\] (25)

M = mass matrix
C = damping matrix
\(C_{cor}\) = Coriolis matrix
K = stiffness matrix
\(K_{spin}\) = Spin softening matrix
F = external forces and the centrifugal forces

Equation 25 is taking in account both the spin softening and the Coriolis effect. Equation 25 is used by Ansys when the displacements are calculated in a rotating reference frame [11].
3.2.2 Stationary reference frame equation

\[
[M]\ddot{u} + [(C) + [C_{\text{gyro}}] \dot{u} + [K]u = \{F\}
\]

Equation 26 is used by Ansys when the displacements are calculated in a stationary reference frame. The gyroscopic terms in the matrix are explained from the kinetic energy expression as shown before in the report in section 2.5. The gyroscopic matrix corresponds to Coriolis and spin softening effect in the Equation 25 describing the rotating reference frame equation [11].

3.3 Units in Ansys

The units in Ansys, like in all finite element programs, are not predefined and therefore the engineer has to choose by herself/himself which kind of unit system he/she wants to use. It is important to use the same units system in the whole model. For this thesis work are SI-units chosen. However, instead of meter, the column of the millimeter system in Table 4 was used in Ansys during the thesis work. Table 4 is showing the SI-units for both m and mm.

Table 4: Table over units in Ansys

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Symbol</th>
<th>SI unit</th>
<th>mm unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td>L</td>
<td>m</td>
<td>mm</td>
</tr>
<tr>
<td>Mass</td>
<td>m</td>
<td>kg</td>
<td>ton</td>
</tr>
<tr>
<td>Force</td>
<td>F</td>
<td>N</td>
<td>N</td>
</tr>
<tr>
<td>Time</td>
<td>t</td>
<td>s</td>
<td>s</td>
</tr>
<tr>
<td>Pressure</td>
<td>p</td>
<td>Pa</td>
<td>MPa</td>
</tr>
<tr>
<td>Velocity</td>
<td>v</td>
<td>m/s</td>
<td>mm/s</td>
</tr>
<tr>
<td>Acceleration</td>
<td>a</td>
<td>m/s²</td>
<td>mm/s²</td>
</tr>
<tr>
<td>Density</td>
<td>ρ</td>
<td>kg/m³</td>
<td>ton/mm³</td>
</tr>
<tr>
<td>Young's module</td>
<td>E</td>
<td>Pa</td>
<td>MPa</td>
</tr>
<tr>
<td>Stress</td>
<td>σ</td>
<td>Pa</td>
<td>MPa</td>
</tr>
<tr>
<td>Frequency</td>
<td>f</td>
<td>Hz</td>
<td>Hz</td>
</tr>
<tr>
<td>Temperature</td>
<td>T</td>
<td>K</td>
<td>K</td>
</tr>
<tr>
<td>Spring constant</td>
<td>k</td>
<td>N/m</td>
<td>N/mm</td>
</tr>
<tr>
<td>Torsion spring constant</td>
<td>k₁</td>
<td>Nm/rad</td>
<td>Nm/rad</td>
</tr>
<tr>
<td>Rotational speed</td>
<td>(Ω)</td>
<td>radians/s</td>
<td>radians/s</td>
</tr>
<tr>
<td>Damping constant</td>
<td>c</td>
<td>kg/s</td>
<td>ton/s</td>
</tr>
<tr>
<td>Radius</td>
<td>r</td>
<td>m</td>
<td>mm</td>
</tr>
<tr>
<td>Displacement</td>
<td>u</td>
<td>m</td>
<td>mm</td>
</tr>
<tr>
<td>Damping torsion constant</td>
<td>c₁</td>
<td>Nms/rad</td>
<td>Nms/rad</td>
</tr>
</tbody>
</table>

3.4 Modal analysis in Ansys

Modal analysis is carried out to find the natural frequencies and the mode shapes of the structure. It is difficult to calculate the right natural frequencies because they are changing with these variables, which is explained in the rest of this subsection. The natural frequencies are changing with the rotational speed because the journal
bearings are dependent by the rotational speed. The properties in a journal bearing that are rotational speed dependent are viscosity, oil temperature and load. To make it even more difficult, the natural frequencies can also change at constant rotational speed due to e.g. changed load, influence of mass unbalance, changed temperature or misalignment of the rotor. The natural frequencies are also changing with the rotational speed because of the gyroscopic moments are affecting the disks. Connection with a shaft or gearbox can change the natural frequencies of the machine due to the extra mass which are connected to the rotor. It is important to consider the whole structure when calculations of rotor dynamics should be done but in this thesis work only the gas turbine rotor was considered [2][5][12].

Prestressed effect is coming from the centrifugal force which trying to expand the disks and the shafts. The dynamic properties can be changed with expanded shafts and disks. To include the prestressed effect a static analysis has to be done before the modal analysis. The static analysis is calculating the stresses and the displacements in the structure which are then used in the modal analyses.

Modal analysis can be done in Ansys in seven ways, Reduced, Block Lanczos, PCG Lanczos, Subspace, Unsymmetric, Damped and QR Damped. What kind of technique to use depends on which kind of structure should be studied [11]. In Appendix A.2 are all seven different methods explained except for the QR Damped method which is explained below.

QR Damped was the method which was chosen for most of the analyses. The method is combined with the good thing about Block Lanczos together with the complex method, Hessenberg method. (More information about the Block Lanczos, QR algorithm and Hessenberg method can be found on www.wikipedia.org.) By using the QR algorithm, a smaller eigenvalue problem can be solved in the modal subspace. The method is calculating the complex eigenvalues and eigenvectors for a damped linear system. It supports unsymmetrical stiffness and damping matrices. A spinning rotor with bearings is producing unsymmetrical matrices and because of that the problems need a solver that can handle unsymmetrical matrices. The QR Damped eigensolver works best for large models because the solver’s accuracy is dependent on the number of nodes. It needs a minimum number of nodes to get a good result. The QR damped method is not recommended for system which is critically damped or overdamped. Ansys is recommending the Damp method for small models, see section A.2.6. For problem where there is a spinning rotor with gyroscopic stiffening effects or a system with damping, the solution from QR Damped will be a complex number s that is showing Equation 27 [11].

\[ s = p \pm i\omega_d \]  

where:
- p = the real part of the complex eigenvalue
- \( \omega_d \) = the imaginary part of the complex eigenvalue (frequency)

The system is stable if the real part is negative and unstable is the real part is positive.
3.5 Harmonic force response analysis in Ansys

Ansys has three different methods to calculate the harmonic force response, the Full method, the Reduced method and the Mode Superposition method. It is only the Full and the Mode Superposition method that are working in a stationary reference frame [11]. Only the Full and the Mode Superposition method are described in the report.

3.5.1 The Full method (Full)

The Full method is the easiest method to use because the engineer doesn’t have to consider any master degree of freedom as in the Reduced method and the mode shape as in the Mode Superposition method see [11]. The method is using the full mass matrix and is not making any approximations. Unsymmetrical matrices can be used in the method and that is good because unsymmetrical matrices are used for bearing problems in a rotor. All types of loads are accepted during the calculation. The problem is that it takes longer time to perform the calculations with this method than with other methods for most of the problems [11].

The Full method is using the same solvers as when a static calculation should be done. There are nine different solvers in the Ansys package and the one chosen for the Full method is the Sparse solver. Sparse solver is a direct solver and not any iterative equation solver. The Sparse solver is solving problems with symmetric and unsymmetrical matrices and bearing problems have unsymmetrical matrices. It is also working well where the elements are poorly shape. The solver is both robust and quick. All these arguments are the reason why the Sparse solver is chosen for the rotor dynamics problems. The problem with the solver is that it needs a lot of memory because it reads in the whole model into the primary memory to get shorter calculation time but that is not any problem when the problem is solved with a server which has a lot of memory. The solver can also be selected to not read in the model to primary memory but the calculation time will be longer when [11].

3.5.2 Sparse solver

A problem can either be solved with an iterative solver or a direct solver. Then a problem is solved with a direct method, the program is doing a Gaussian elimination to solve the unknown vector of displacement \( \{u\} \) in Equation 28. The equations 28 to 31 are from [11].

\[
[K]\{u\} = \{F\} \tag{28}
\]

where:
- \( [K] \) = stiffness matrix
- \( \{u\} \) = global vector of unknown displacements
- \( \{F\} \) = global force vector

The direct method is a process which involving factorizations of the matrix \( [K] \) into lower and upper triangular matrices \( [K] = [L][U] \). Equation 28 gets like this after the factorization.

\[
[L][U]\{u\} = \{F\} \tag{29}
\]
Where:
\([L]\) = lower triangular matrix
\([U]\) = upper triangular matrix

\([U]\{u\}\) in Equation 29 is when substituted with \({w}\), which can be solved by a forward operation in Equation 30. After the forward operation, \(\{u\}\) can be solved by back substitution with the upper triangular matrix in Equation 31.

\[
\begin{align*}
[L]\{w\} &= \{F\} \\
[U]\{u\} &= \{w\}
\end{align*}
\] (30)

(31)

3.5.3 The Mode Superposition method (MSUP)

The Mode Superposition is one of the methods which could be used for harmonic force response analyses. The method is using a modal analysis before the harmonic analysis to get the mode shapes of the modes. The Mode Superposition is using the result from the modal analysis and adds together the eigenvectors (the mode shapes) to calculate the harmonic force response see [11]. That makes this method much faster than The Full method and The Reduced method for most of all the problems. By using the mode shapes the method knows where the responses are going to be and can make better calculation at these frequencies. The peaks at the critical speeds will be more accurate and smoother because the method is making the step length shorter at the peaks. The element loads which were applied in the modal analysis can be applied in the harmonic force response analysis by using the command lscale. The element loads can be different temperatures. Prestresses effect can be added into the analysis, when a static analysis is made before the modal analysis to get all the stresses in the structure. This is the same procedure which is done for only a prestressed modal analysis [11].

3.5.4 Chosen method for the harmonic response analyses in Ansys

Because of the rotational speed is not constant the Full method going to be used. The Mode Superposition is faster than the Full method when the rotational speed is constant. The modes are changing with the speed because of the gyroscopic stiffening effect and because of that the Mode Superposition method has to do a new modal analysis for every step and that makes the calculation time longer. If the modes were not changing the MSUP method only has to do one modal analysis and can when use the mode shapes for all the steps. Therefore the Full method is faster on problem where the modes are changing for every step. There are some restricting with both of these methods. The first is that all the loads have to be sinusoidal and the frequency of all the loads has to be the same. The second one is that no nonlinearities are allowed [11].

3.6 Damping in Ansys

There are a lot of different ways damping can be introduced to the model in Ansys. The response will theoretically go to infinity if there are no damping in the system, thus damping has to be introduced. It is difficult to know the right damping constants for the model and therefore only the element damping is taken in account in the model.
Element damping is added from elements like Combi214 or Combin14 which have damping properties inside the element. These two elements are important during modeling a rotor. The damping and spring coefficients are set with a real constant in Ansys. The coefficients don’t need to be constant since they can be referred to a table where the coefficients are changing with the rotational speed. It is normal that the properties of the bearings are changing with rotational speed.

The rest of the damping constants are explained in Appendix A.3 because it is good to know that there are many different ways damping can be included into a model [11].

### 3.7 Mass unbalance forces

In all manufactured gas turbines there are mass unbalances due to tolerances and geometries. The equation of how the mass unbalance force is changing with the rotational speed in a 2D problem like in the Cases three to five in section 4.3-4.5, is explained below in Equation 32 [11].

\[
\{ F(t) \} = \begin{cases} 
F_1 \\
0 \\
0 \\
0 
\end{cases} \cos(\Omega t) + \begin{cases} 
0 \\
F_2 \\
0 \\
0 
\end{cases} \sin(\Omega t)
\]  

(32)

Then mass unbalance forces should be used in Ansys, they are set by these commands.

- \( \{ F_1 \} = m_u e \Omega^2 \) = The real force (input Value under \( F \) command in Ansys)
- \( \{ F_2 \} = m_u e \Omega^2 \) = The imaginary force (input Value2 under \( F \) command in Ansys)

where:
- \( m_u \) = the mass of the unbalance in the system
- \( e \) = distance from the axis of rotation to the mass unbalance
- \( \Omega \) = rotational speed

The solutions from a harmonic force response analysis can either be set to real and imaginary or amplitude and phase shift. When using real and imaginary as outputs the solutions will be \( u_1 \) (the real value) and \( u_2 \) (the imaginary value), but when using the amplitude and phase shift as output the solutions will as Equation 33 and 34.

\[
\begin{align*}
\max u & = \sqrt{u_1^2 + u_2^2} \\
\Phi & = \arctan \frac{u_2}{u_1}
\end{align*}
\]  

(33)  

(34)
4 Verification of Ansys with simple models

To verify Ansys five simple cases were made and they are explained in section 4. Ansys are both compared to theoretical calculations and Ardas.

4.1 Case one

It is an easy example of a bar with diameter 200 mm and length 2000 mm. The idea is just to look at the first lateral mode and compare it to an analytical formula.

First was the beam calculated analytically with the Equation 35 from [5]. The equation is for the first lateral frequency for a bar with fixed ends. The results are shown in Table 5.

\[ f = \frac{\pi}{2} \sqrt{\frac{EI}{mL^3}} \]  

(35)

\( E = 211 \times 10^9 \) Pa  
Mass of the beam \((m) = 493.23 \) kg  
Length of the beam \((L) = 2 \) m  
Second moment of area \((I) = 7.8539 \times 10^{-5} \) m\(^4\)  
\( \rho = 7850 \) kg/m\(^3\)  
\( \nu = 0.3 \)

In Ardas was the geometry and material data declared with the both ends rigidly fixed.

In Ansys was the geometry meshed with Solid186 which is a 20 nodes brick element. Both ends are fixed in all directions.
Table 5: Results Case one

<table>
<thead>
<tr>
<th>Frequency (Hz)</th>
<th>Ansys</th>
<th>Ardas</th>
<th>Analytical</th>
</tr>
</thead>
<tbody>
<tr>
<td>100.5</td>
<td>101.2</td>
<td>101.8</td>
<td></td>
</tr>
</tbody>
</table>

The frequencies are close to each other.

4.2 Case two: Verification between Ansys and Ardas with bearings

The second verification is with bearings at the ends of the bar. This is more realistic than the first one. The spring constants for the bearings are set to 5*10^7 N/m. In Ansys a special element is used which is called Combi214 to simulate a bearing. Combi214 is a new element in Ansys v11 and developed for rotor dynamics applications. The element has springs and dampers in four directions; one horizontal, one vertical and two cross coupled springs. In Figure 12 is the element shown [11][12].

The material data was the same as in Case one. All the numbers are from forward whirling. The difference between forward and backward whirling is very small because the rotor doesn’t have any disk that gets influenced by the gyroscopic stiffening effect.

Table 6: Result Case two

<table>
<thead>
<tr>
<th></th>
<th>Ansys</th>
<th>Ardas</th>
</tr>
</thead>
<tbody>
<tr>
<td>The first mode (Hz)</td>
<td>59.5</td>
<td>59.8</td>
</tr>
<tr>
<td>The second mode (Hz)</td>
<td>120.3</td>
<td>120.8</td>
</tr>
</tbody>
</table>

Also here the natural frequencies are close to each other.
4.3 Case three: Verification between Ansys and analytical model

A simple rotor is used for this case shown in Figure 13. The rotor has a disk, a shaft and is rigidly supported at the shaft end. The disk is hanging free without any support in this case. The mass center of the disk is not coinciding with the axis of rotation, it is offset with $e = 0.001$ m away from the axis of rotation, shown in Figure 14.

Figure 13: Case three model
A sketch over the rotor with overhung disk. It is rigidly supported on the left side

Figure 14: Mass center
The mass center is offset on the disk
4.3.1 Analytical model for Case three
In the analytical model will the torsional modes not going to be considered. The motion of the rotor is described with four degrees of freedom, two lateral and two rotational. \( \{u\} = \{y \quad z \quad \psi \quad \theta\}^T. \)

The Equation of motion is described in Equation 36.

\[
[M]\ddot{\{u\}} + [C_{\text{gyro}}]\dot{\{u\}} + [K]\{u\} = \{F(t)\}
\]  

(36)

Where:

\[
[M] = 
\begin{bmatrix}
 m & 0 & 0 & 0 \\
 0 & m & 0 & 0 \\
 0 & 0 & J_T & 0 \\
 0 & 0 & 0 & J_T
\end{bmatrix}, \quad [C_{\text{gyro}}] = 
\begin{bmatrix}
 0 & 0 & 0 & 0 \\
 0 & 0 & 0 & \Omega J_p \\
 0 & 0 & -\Omega J_p & 0 \\
 \end{bmatrix}
\]

\[
[K] = 
\begin{bmatrix}
 k_{11} & 0 & 0 & -k_{12} \\
 0 & k_{11} & k_{12} & 0 \\
 0 & k_{12} & k_{22} & 0 \\
 -k_{12} & 0 & 0 & k_{22}
\end{bmatrix}
\]

\[
I = \frac{\pi d^4}{64}, \quad J_p = \frac{mD^2}{8}, \quad J_T = \frac{mD^2}{16}, \quad k_{11} = \frac{12EI}{L^3}, \quad k_{12} = \frac{6EI}{L^2}, \quad k_{22} = \frac{4EI}{L}
\]

\([C_{\text{gyro}}]\) is the gyroscopic matrix that is important during an analysis of a rotor [14]. To do an analytical calculation of the eigenvalues and the harmonic force response, Matlab codes had to be written. The codes can be seen in Appendix A.5.7-A.5.9 for Case three, four and five.

4.3.2 Ansys model for Case three
The model is built like a 3D model in Ansys and by using the Coriolis command will the program automatically consider the gyroscopic effect. The rotational speed is set
by the \textit{cmomega} command. A stationary reference frame is used because it is the frame Ansys recommend for drawing a Campbell diagram. The codes for the model can be seen in Appendix A.5.1-A.5.6 for Case three, four and five.

4.3.3 The Modal Analysis for Case three

A modal analysis is done both with Ansys and the analytical model. The modes are printed in the same Campbell diagram, Figure 15, the forward and the backward whirling can be easily seen because the modes are separating into two lines, one decrease (backward) and the other one increase (forward). From Ansys you get also the torsion mode which is a straight line in Figure 15. The two first lateral mode lines are not separating so much from each other. This is because the disk doesn’t have any transversal rotation which is needed in order to get an influence of the gyroscopic effect. In the two highest lateral mode lines, the gyroscopic effect is strong, and the lines are separating a lot from each other. The excitation line is coming from the rotational speed of the rotor. The result from the Ansys and the analytical model is the same.

![Campbell diagram over third case](image)

\textit{Figure 15: Campbell diagram from a modal analysis for Case three}

\textit{The dashed lines are from Ansys. The legend is over the diagram}
There will be a force acting on the disk, because of the mass center is not coinciding with the axis of rotation. The mass unbalance will be the mass of the disk, \( m_u = m \). The force equation is shown in Equation 37.

\[
\{ F(t) \} = \begin{bmatrix} m_u e \Omega^2 \\ 0 \\ 0 \end{bmatrix} \cos(\Omega t) + \begin{bmatrix} 0 \\ m_u e \Omega^2 \\ 0 \end{bmatrix} \sin(\Omega t)
\]

(37)

This force is put into both of the Ansys model and the theoretical model.

In Ansys was The Full method used for this harmonic response analysis, and as in the modal analysis were the *coriolis* and *cmomega* commands used to get the gyroscopic effect and the rotational speed. The results from Ansys and the analytical model are very similar, shown in Figure 16. The peak is at the same rotational speed as where the excitation line crosses the mode lines, about 125 rpm and 150 rpm in Figure 15, and that is called the critical speed. This is the reason why the rotational speed should not be too close to any position where the excitation line crosses any of the mode lines.

---

**Figure 16: Response analysis for Case three**

*The dashed line is from Ansys*
4.4 Case four: Verification between Ansys and analytical model with bearing without damping

Fourth case is almost the same as the third case except there are two springs connected at the center of the disk, where the vertical spring is twice as stiff as the horizontal spring. The stiffness of the springs $k_y = 3 \times 10^5$ N/m and $k_z = 6 \times 10^5$ N/m.

4.4.1 Analytical and Ansys model for Case four

To get into the spring constants into the analytical model is easy, only add the spring constants into the stiffness matrix $[K]$. [Equation]

$$[K] = \begin{bmatrix} k_{11} & 0 & 0 & -k_{12} \\ 0 & k_{11} & k_{12} & 0 \\ 0 & k_{12} & k_{22} & 0 \\ -k_{12} & 0 & 0 & k_{22} \end{bmatrix} + \begin{bmatrix} k_y & 0 & 0 & 0 \\ 0 & k_z & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

The analyses in Ansys done by using element Combi214 as the bearing and the spring constants can easily be added into the model. The rest is done in the same way as in Case three.

4.4.2 Modal Analysis for Case four

In the Campbell diagram in Figure 17 have the two first lateral modes moved upwards and they have also been separate from each other. This is because the stiffness of the spring in vertical direction is twice as stiff as in horizontal direction. The third and fourth lateral modes have not moved anything from Case three and that is because the disk in these modes isn’t moving in y- or z-direction only rotating. The result from Ansys and the analytical model is the same.
4.4.4 Harroidic force response analysis for Case four

In Case three where no springs added to the model but in Case four there are springs connected to the disk. The spring constant is twice as stiff in vertical direction as in horizontal direction and that means that the disk is going move in an elliptic curve. The ellipse will have a major axis (longest one) “A” and minor axis (shorter one) “B”. It is the major axis and minor axis that are printed out in graph below. The response force is the same as in Case three. Also here are results from Ansys very similar to analytical model, shown in Figure 18. When the minor axis is less than zero it will be a backward whirl and if minor axis is larger than zero it will be a forward whirl.
4.5 Case five: Verification between Ansys and analytical model with damping

Case five is almost like Case three and four except now there are also two dampers in horizontal and vertical direction connected to the center of the disk next to the springs. The damper constants are \( c_y = 500 \text{ Ns/m} \) and \( c_z = 250 \text{ Ns/m} \).

4.5.1 Analytical and Ansys model for Case five

In Analytical model the damping constants are added together with the gyroscopic matrix [14].

\[
[C] = [C_{gyro}] + [C] = \begin{bmatrix}
0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 \\
0 & 0 & 0 & \Omega J_p \\
0 & 0 & -\Omega J_p & 0
\end{bmatrix} + \begin{bmatrix}
c_y & 0 & 0 & 0 \\
0 & c_z & 0 & 0 \\
0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0
\end{bmatrix}
\]

The stiffness matrix is the same as from Case four.

In Ansys the damping constants are added to the model by setting the real constant for Combi214. The real constant is setting the properties of the bearings.
4.5.3 Modal analysis for Case five
The modal analysis is done in the same way as in Case four and Ansys is showing good agreement with the analytical model and shown in Figure 19.

![Campbell diagram over fifth case](image)

*Figure 19: Campbell diagram from a modal analysis for Case five. The dashed lines are from Ansys. The legend is over the diagram*.

4.5.4 Harmonic force response analysis for Case five
The movement of the disk will be an elliptic curve as in Case four. Ansys is showing good agreement with the analytical model and shown in Figure 20.
Figure 20: Response analysis for Case five
5. Modeling of SIT’s gas turbine rotor

The goal in this thesis work was to model one of SIT’s gas turbine rotor with the compressor and the turbine disks. The rotor should then be analyzed to see if the results are reliable. The model which was used came from a CAD file formed as an Iges file. The CAD model had only the lines over the rotor and no areas. First thing that had to be done was to simplify the model and take away the small lines. This was done to making the meshing easier and to get fewer elements. The simplified wireframe model was then taken into Ansys where it was modified a little bit more to be able to produce areas of the lines. The model was divided into many areas to make it easier to mesh it. The area model was then meshed with an element which is not solving anything. It is called Mesh200 and is in the Ansys package. The areas had to be meshed before it could be rotated so the program understands that also the elements should be rotated. To make a well shaped area mesh is important to get the right result in the end. It is also good not to get too many elements in the 3D model because that makes the model heavier to analyze. The meshed 2D model was rotated 360 degrees to produce the 3D solid model of the rotor which should be studied, shown in Figure 21.

![Figure 21: Ansys model of the gas turbine rotor](image)

5.1 Element

The element that the rotor is built of is Solid186 which is a 20 nodes brick element. The element is good because it takes in account the gyroscopic matrix which is needed in rotor dynamics. The element has three DOF (x, y and z) for every node. Solid186 was chosen because it had mid nodes at the edge which is good when curved lines should be modeled. A stationary reference frame is also supporting this element.
It is also easy to rotate brick elements which were needed to do to make the 3D model. The problem with Solid186 is that it doesn’t have any rotational DOF, and therefore graphs of the torsional modes can not be drawn. This was solved by a thin beam at axis of rotation, and the beam nodes were connected by constraints with the nodes in solid elements.

Beam188 was chosen to be the beam element to the thin beam at axis of rotation, because it is easy to work with it and no real constants need to be declared. A cross section is used instead and Ansys has already sections made of how the beam will look like. The mass of the beam and the young’s module were set to very low values so the beam wouldn’t disturb the rotor. Beam63 was chosen to be the beam element to the blades, it works in the same way as Beam188 except Beam63 elements don’t have the gyroscopic matrix. The beam elements can only have the gyroscopic matrix included when they are coinciding with the axis of rotation and the blade beams are not located at the axis of rotation. That is the reason why Beam63 was chosen to the blades.

Combi214 was chosen as the bearing element because it has springs and dampers in four different direction y, z, yz and zy. That is needed when a journal bearing should be modeled. The spring and damper constants were declared by real constants in Ansys [11].

One model was done with 16000 elements, the rotor became stiffer and the critical speeds became 5 % higher. This is showing that the model needs to have at least a minimum of elements to get the right result. The final model has about 24 000 elements but with 38000 elements has been tested. The result was the same, and therefore it was concluded that 24000 elements are good enough.

5.2 Bearing

The properties of the bearings were calculated in a program which has the solver ALP3T. The bearing calculation was done by Thomas Domeij at GRCRM on SIT AB in Finspång [2]. The bearing properties were printed out into two tables, one for each bearing. The bearings in the gas turbine are of the tilting-pad type and it can be seen in the diagrams below that the cross couplings stiffness are much lower than the vertical and horizontal stiffness. This is the best thing with tilting-pad type because the stability of the rotor is better with low cross coupling stiffness. The stiffness of the bearings is decreasing very much with increasing rotational speed. The decrease of the bearing stiffness is one of the reasons that make rotor dynamics calculations difficult. The bearing stiffness and damping is shown in figures 22-25. Bearing 1 is located on the compressor side and bearing 2 is located next to the turbine disks.
Figure 22: Stiffness curve for bearing 1
(Cyan and blue is the same line)

Figure 23: Stiffness curve for bearing 2
(Blue and cyan is the same line)
Figure 24: Damping curve for bearing 1
(Red and black is the same line, also blue and cyan)

Figure 25: Damping curve for bearing 2
(Blue and cyan is the same line)
Both tables were imported into Ansys as two tables and by interpolating in the tables the program can pick out the right stiffness and damping constants for the given rotational speed. The real constants for the element Combi214, which is the element for the bearings, are modified every time the rotational speed is changing. The real constant command can only add six constants at once and that means that command rmore needs to be used for adding the last two damping constants. The damping in the bearings is the only damping which will be added to the model. There are many possibilities to include damping with Ansys Classic but the problem is to actually know how large the damping constants are for the different damping parts, for example the material damping. The material damping is at least much smaller than the bearing damping so it can be ignored anyway. Thus, there are a lot of possibilities with Ansys to include damping to the model when the damping constants are known.

5.3 Blades
One of the biggest problems was to get the blades into the model without making the model too heavy to solve. Two different methods have been evaluated. One where the rings were made of small Solid186 elements connected to the rotor with dummy Solid186 elements. The dummy elements didn’t have any mass and no strength in tangential direction but very strong in axial and radial direction. In the second method beam elements were used instead. The beam elements, Beam 63 elements, were made as rings around the rotor and were connected to the rotor by a lot of constraints. The weights of the blades were added at the blade’s center of gravity. The blades at each disk have different masses so the density of every ring had to be set different, so the weight of every ring is the same as the blades at that disk. Cylindrical coordinates were used instead of a global Cartesian coordinate system to be able to set the tangential strength to almost zero for the Solid186 element rings. The radial and axial strength was set to a very high value to prevent that only the rings should vibrate. When using a cylindrical coordinate system, the x-direction is the radial direction, the y-direction is the tangential direction and the z-direction is the axial direction.

5.4 Modal analysis
Two different modal analyses have to be done when dealing with rotor dynamics, they are undamped critical speed analysis and damped critical speed analysis. Undamped modal analysis is done to get an idea what the mode shapes look like. It is important to know how the mode shapes are because the engineer needs to know where the unbalance masses should be placed in a harmonic force response analysis. The mass unbalance forces should to be located on the places at the rotor where the deflection is largest. If the mass unbalance force is located on the place where the largest deflection is so can a special mode be excited during a harmonic force response analysis and the response curve will show the greatest peak at the mode’s critical speed. The greatest peaks will be on different places depending on where the unbalance force is placed. Undamped modal analysis should be done with different bearing stiffness to see how the modes are changing with the support stiffness. It is important to understand that the critical speeds that are calculated from the undamped modal analysis aren’t the same as the correct critical speeds, only estimations [12]. The mode shapes of the first three lateral modes can be seen in Figure 27, Figure 28 and Figure 29. These results were used in the harmonic force response analyses when
the mass unbalance forces should be placed. A 2D model of rotor is shown in Figure 26.

**Figure 26: 2D model of the rotor**

**Figure 27: First lateral mode at an undamped modal analysis**

**Figure 28: Second lateral mode at an undamped modal analysis**
The second analysis, that an engineer has to do, is damped critical speed analysis and that is when the damping and the right support stiffness are included into the system. When using damping the mode frequencies from the undamped modal analysis are changed and they are corresponding to the real critical speeds. The modal analysis should be done from 0 % to 125 % of the operating velocity [12]. In this thesis the critical speeds were analyzed from 2500 rpm to 20000 rpm.

The prestressed effects were also analyzed to see if it has any influences on the critical speeds. Prestressed analysis is performed by doing a static analysis before the modal analysis so all the stresses and displacements, which are created by the centrifugal force, are included into the modal analysis. All the modal analyses were done with the QR Damped method because it works best with big models and damping can be included.

5.5 Harmonic force response analysis

A harmonic force response analysis is important to perform to make sure that the rotor never touches the bearings. Response analyses are done to prevent large vibrations because it may lead to fatigue and vibration problems. A response analysis is showing how much the rotor is deflected from the axis of rotation. To get a response mass unbalance forces have to be added to the model. Where to place the mass unbalance forces are decided by an undamped modal analysis as discussed above in section 5.4. The mass should be placed where the deflections of the modes are the largest [12]. The mode shapes of the SIT AB gas turbine are shown in Figure 27, Figure 28 and Figure 29. Thanks to these figures can the mass unbalances be located on places shown in Figure 30.
A. The first analysis is when the mass unbalance force is on the turbine blade and thanks to that the second modes will be excited in the response curve.
B. In second analysis the mass unbalance is placed in the middle of the bearings and that is exciting the first modes.
C. In the third analysis are the mass unbalances placed in two places. The places are one-quarter and three-quarters of the distance between the bearings and they are placed opposite of each other (180 degrees out of phase of each other). The mass unbalance forces’ position will excite the third modes in the response curve [12].

The method that was chosen for the harmonic force response analyses in Ansys was the Full method and it was done with 134 steps from 1500 rpm to 11500 rpm.

![Figure 30: Mass unbalance](image)
The balls are showing where the unbalances are in the three different response analyses

### 5.6 Imposing temperatures into the model

A gas turbine is not operating at a constant temperature. The temperature is changing along the rotor. The higher temperatures make the material weaker and the rotor weaker, therefore will the natural frequencies decrease. The temperatures of the rotor were given by Thomas Domeij [2]. The different temperatures were included into the Ansys model and the damped critical speeds were analyzed. To include the temperatures are not difficult in Ansys but it would be better if the model was divided into several volumes where every volume only had one temperature instead of several. It is good to know when a new model should be set up. A damped critical speed analysis with the imposed temperatures was compared with a damped critical
speed analysis without the imposed temperatures. The critical speeds are 3-5 % lower with the imposed temperatures. The torsion modes are also reduced by 3-4 %.

5.7 Static analysis
A static analysis is done to see if the model in Ansys has the same stiffness as the model in Ardas. The deflections were the same in the two programs. Static analyses are done to see how much the rotor deflects by its own weight. The gravity constant is included into the model. Gravity constant is normally never included in rotor dynamics calculation because the deflection, done by the gravity, of the rotor is already taken in account as the normal state. All the other deflections, which are done for example by mass unbalance, are measured from that normal state and because of that can the gravity constant be ignored. It can be compared with a mass hanging in a spring, where the gravity constant is also ignored by the same reason.

5.8 Result of the modal analyses
Modal analyses were done with different models to see if the results are satisfying. The results were compared with Ardas to see how well the results agreed. Two different models with blades have been tested; where one had blades made of beam elements and the other had blades made of Solid186 elements. It is only the Solid186 elements that are including the gyroscopic matrix into the calculation. The results are showing that the gyroscopic stiffening effects are slightly larger in the Solid186 blade model than the beam blade model and that can be seen when the Solid186 model’s backward whirling critical speeds are little bit lower and the forward whirling critical speeds are slightly higher compared to the beam model. The mode lines have been separated more in the Solid186 elements model than the beam model and that result was expected.

The results from the Solid186 blade model are compared with Ardas and all the lateral critical speeds are lower than Ardas. The four first lateral critical speeds are only 3-8 % lower than Ardas and the next three lateral critical speeds are 10-20% lower. The biggest differences, between Ansys and Ardas, are in a backward whirling motion and in the situation where the bearings are involved a lot in the motion. This was confirmed by an undamped critical analysis without any bearings and the critical speeds were much closer to Ardas. The critical speeds for the forward whirling modes were almost the same as Ardas, only 1-2 % difference. The two first back whirling modes were 8-10 % difference from Ardas. The third backward whirling mode had the same critical speed as Ardas. The problem is that the connections with the turbine disks are too weak compared to Ardas. The Ansys model is made with a gap at the connection between the coupling and the first turbine disk like as in the drawings, but the Ardas model doesn’t have that gap. A modified version was made without the gap and that model came closer to the Ardas model with only 6-7 % difference.

The torsion mode’s frequencies are differing between the Solid186 blade model and the beam blade model, where first torsion mode in the Solid186 blade model is 7 Hz lower than the beam blade model. The Solid186 blade model’s torsion modes have the same critical speeds as Ardas.
Prestressed analyses have been done but the results are not satisfying. The difference in critical speeds between a prestressed rotor and a normal rotor are too big up to 800 rpm. Ansys gets also always problem to sort the prestressed modes in the Campbell diagram. The torsion mode that should be almost constant over the rotational speed range is changing a lot and therefore isn’t the prestressed function reliable. During static analyses with constant rotational speed the largest radial expansions were 0.1 mm at 5000 rpm and 0.4 mm at 10000 rpm. This can be compared with the thermal radial expansion which is about 1 mm and that is ignored during rotor dynamic calculations [2]. The expansions over the weak parts, which are twisting areas in the torsion modes, are even smaller and can be neglected, and therefore shouldn’t the torsion frequency be increased with 75 Hz between zero rpm and 10000 rpm. Thus, prestressed analyses are not recommended because it is showing odd results and can be confusing.

5.9 Result from the harmonic force response analysis
The harmonic force response curves are very similar to the response curves from Ardas and the critical speeds that were calculated from the damped critical speed analyses above are at the peaks. That was expected but it is always good to see. The peaks are slightly higher in the Ansys analysis than in the Ardas analysis. The analysis of a model with 24000 elements in 134 steps with the step length 75 rpm took over 15 hours.

5.10 Orbit plots for gas turbine rotor
As discussed above in section 5.1, the disadvantage with the solid element Solid186 is that it can not draw orbits in Ansys because it doesn’t have any rotation DOF. The problem was solved by adding an extra beam at the axis of rotation where the beam nodes were connected to the solid elements by constraints. Thanks to this beam orbits can be drawn and compared with Ardas. The orbits plots are similar with Ardas. The orbit plots are good to have to compare with other programs because they are showing the motion of the rotor.

5.11 Bearing connections
The problem in all the analyses with the bearings is the connection between the bearing nodes and the model. It wouldn’t be problem if the rotor was solid on the location where the bearings are placed but the gas turbine rotor isn’t solid, therefore the engineer needs to use constraints to connect the bearing nodes to the model. To get the right connections between the bearing nodes and the rotor are the difficult part because if the connections are made wrong the critical speeds can be wrong. It requires a lot of knowledge to really know how the connections should be done because the critical speeds can vary with different connections.

5.12 Sources of errors
The Ansys model is not showing the same result as Ardas. The difference can be because of different methods were used, Ansys is using Finite Element method which is working in 3D and Ardas is using Transfer Matrix method which is working in 1D. None of the methods are totally exact because both of them are numerical methods which always have errors but the difference should not be 10 % as difference between the solutions. The model has a very complex geometry, and therefore the geometry was maybe a little different between the Ansys model and the Ardas model. The
modeling of the bearing connections in Ansys can maybe be better because it is difficult to get a good connection when the model isn’t solid. How the blades are modeled can maybe be better because they are affecting the dynamics a lot when they are located on the top of disks. The author, which has been evaluating Ansys, doesn’t have any long experience of using Ansys and SIT AB doesn’t have either any experience of using Ansys in rotor dynamics applications. Thus, the calculations and modeling aren’t maybe done in the best way because of that.
6 Conclusions

The results from the simple models were very reliable and the results are the same as the theoretical results in all the cases. The gyroscopic matrix with the Solid186 elements is working in Ansys. Most of the examples which had been done before by others are made with beam elements, not with solid elements. Therefore, it is good to show that the rotor dynamics applications are at least working for simple models which are made of solid elements.

One of problems was to know how to model the blades in the Ansys model because the blades have a large influence of the dynamic when they are located on the disk. The two different blade models that are evaluated in this thesis work are not showing the same result. The best model is the Solid186 blade model because it takes into account the gyroscopic effects also in the blade and the torsion modes has the same critical speeds as the model from Ardas, where the beam blade model has a mode which is 7 Hz higher. The biggest difference between Ansys and Ardas is in the situation where there is backward whirling motion and the bearings are involved a lot in the motion. That means that the model is either too heavy over the bearings or that the connections with the bearings are made in a bad way.

The prestressed function isn’t working correctly in Ansys. The differences get too big compared with a rotor without the prestresses. The biggest difference is with the torsion modes which are increasing a lot with the rotational speed and they have no reason to increase when they should be almost constant. The prestressed function isn’t recommended to use when the results are not satisfying and they can be confusing.

Imposing the temperatures is lowering the critical speeds with 3-4 %. Ansys has a great system to include the temperatures to the elements in the model.

The harmonic force response curves are very similar to the Ardas curves except that the peaks in Ansys are at lower rotational speeds such as the critical speeds which are also lower in Ansys than in Ardas. The response analysis is working in Ansys but an analysis can take over 15 hours.

Ansys rotor dynamics application isn’t working perfectly yet, there are some problems with commands like prestressed and synchro (command for response analyses which is used to synchronize the frequency on the mass unbalance force with the rotational speed). The critical speeds that are calculated from Ansys are difference than the critical speeds from Ardas. The calculation time is much longer compared to Ardas and a harmonic force response analysis can take over 15 hours. Ansys has some benefits such that the mode shapes can easily be shown visually. More complex calculation can be made, for example the modal analysis can be connected by a thermal analysis to get the right temperatures in the model. The temperatures in Ansys can also differ in the radial direction and not only in the axial direction. The modes are automatically sorted in a Campbell diagram. The opportunities to do more complex calculation are bigger in Ansys than in Ardas but the engineer needs to have a lot of knowledge to use them since otherwise the solutions can be wrong. The problems with the commands and the memory will probably be solved in the future since the rotor dynamics applications are pretty young in Ansys. The rotor dynamics application hasn’t been used so much with solid elements yet, only with beam element
and it is therefore perhaps Ansys is not working perfectly with solid elements yet. However, today Ansys is a lot more capable to do rotor dynamics calculations than it was a few years ago.

There is a potential to use Ansys in the future as an extra analysis tool for more complex analyses and simulations. The rotor dynamics applications are working for simple models, and therefore it should work for more complex structures. The differences between Ardas and Ansys are mainly coming from how models were modeled in Ansys and which kind of approximation that was used. The torsion mode’s frequencies from the model, which is using the Solid186 elements as blades, are the same as Ardas and therefore the model with Solid186 elements as blades are the best model in Ansys. Ansys and Ardas are compared with each other in Table 7.

Table 7: Ansys and Ardas are compared with each other

<table>
<thead>
<tr>
<th>Advantage</th>
<th>Disadvantage</th>
<th>Advantage</th>
<th>Disadvantage</th>
</tr>
</thead>
<tbody>
<tr>
<td>More complex calculations and simulations can be done</td>
<td>Long calculation time</td>
<td>Short calculation time</td>
<td>The calculations and simulations are restricted to 1D</td>
</tr>
<tr>
<td>Different analyses can be combined</td>
<td>Has not been tested so much</td>
<td>Has been tested a lot over the years</td>
<td>Torsion, longitudinal and lateral modes are analyzed separately</td>
</tr>
<tr>
<td>The modes are automatically sorted in a Campbell diagram</td>
<td>Needs a lot of knowledge to do analysis and the meshing are very important</td>
<td>The blades can easily be included as masses above the disks</td>
<td>Every model has to be drawn in Ardas, cad models can not be imported</td>
</tr>
<tr>
<td>The mode shape can easily be seen visually</td>
<td>The bearing connections are difficult get right</td>
<td>Easy to include bearings</td>
<td>The modes are not automatically sorted in a Campbell diagram</td>
</tr>
<tr>
<td>Torsional, longitudinal and lateral modes are always analyzed together</td>
<td>The blades can not easily be included into the model.</td>
<td>Specialized on only rotor dynamics calculations that makes it easier to use</td>
<td></td>
</tr>
<tr>
<td>Cad models can be imported directly</td>
<td>Can get memory problem when Campbell diagram should be drawn</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

7 Problem with running Ansys in the hardware

During the thesis work there were some problems and one thing was a memory problem with the computer. The computer, which was used, was a 32-bits computer that could only release 2 GB memory which left about 1.3 GB to Ansys. That was not enough when the Campbell diagram should be drawn in Ansys. When the Campbell
The gas turbine rotor model had many elements and therefore was it easy to get a too big result file. To solve the memory problem, all the modal analyses were divided into at least two steps to be able to draw the Campbell diagrams. That is the reason why all the modes are not in the same diagram. The problem could be solved if a 64-bits computer is used instead because it can release 4 GB memory instead. If rotor dynamics calculation should be done in Ansys in the future only a 64-bits computer should be used.

8 The future
In the future transient analysis should be carried out and the stator should be added to the model. More understanding regarding the support connection with the bearings is needed and more models should be tested to see how to get the best results. More knowledge about how the blades can be modeled is to be gathered. The blades dynamics can maybe change the modes of the rotor. In my model the blades are only modeled like masses outside the disks. Maybe, together with Ansys, the prestressed function can be improved and maybe another way to produce the Campbell diagram in Ansys can be develop, such that the program doesn’t get any memory problems
Reference list


[2] Domeij, Thomas, GRCRM, SIT AB, private conversations


A.1 Forward whirling versus backward whirling

The whirling motion can either be circular or elliptic. The elliptic motion can be presented by two circles that have the radius $x_p$ for the progressive motion (forward) and $x_r$ for the retrograde motion (backward). The elliptic curve can also be presented as one major axis “A” and one minor axis “B”. The major axis is always positive and the minor axis is positive when there is a forward whirling. The minor axis is negative when there is a backward whirling. The major axis and minor axis can be written like in Equations 38 and 39 \[7\]. Equations 38 to 57 are from \[7\].

\[a = |x_p| + |x_r|\] (38)
\[b = |x_p| - |x_r|\] (39)

The motion is progressive (forward) when $|x_p| > |x_r|$ and retrograde (backward) when $|x_p| < |x_r|$ \[7\].

\[y = x_p + x_r\] (40)
\[z = -ix_p + ix_r\] (41)
\[x_p = \frac{1}{2}(y + iz)\] (42)
\[x_r = \frac{1}{2}(y - iz)\] (43)

Equations 42 and 43 are showing how progressive and retrograde constants are declared for transversal motion. It is declared in same with transversal rotation of the disk in Equation 44 and 45.

\[\theta_p = \frac{1}{2}(\psi + i\theta)\] (44)
\[\theta_r = \frac{1}{2}(\psi - i\theta)\] (45)

The disk from the single disk rotor (Jeffcott rotor) is mounted little skew with an angle, therefore the rotating variables are replaced (Equation 46 and 47). This is done to get influence of the gyroscopic effect into the calculation because the motion needs to be conical to get the influence of the gyroscopic effect.

\[\theta \rightarrow \theta + |\gamma|\cos(\Omega t + \phi)\] (46)
\[\psi \rightarrow \psi + |\gamma|\sin(\Omega t + \phi)\] (47)

The Equations 13 from the section 2.5 get like these with the replaced variables, Equation 46 and 47, in Equation 44 and 45. The damping is set to zero.

\[\ddot{J}_T \theta + \Omega J_p \psi + k_p \theta = -\Omega^2 \left(J_p - J_T\right) |\gamma|\cos(\Omega t + \phi)\] (48)
\[ J_T \ddot{\psi} + \Omega J_p \dot{\psi} + k_\psi \psi = -\Omega^2(J_p - J_T)\psi |\sin(\Omega t + \varphi) \]  

(49)

The skew angle which the disk had been mounted at can be written in complex form in Equation 50.

\[ \gamma = \gamma_c + i\gamma_s = |\gamma| |\cos \varphi + i|\gamma| \sin \varphi \]  

(50)

By setting together equations 44, 45, 48, 49 and 50, two Equations of motion 51 and 52 can be written.

\[ J_T \ddot{\theta}_p - i\Omega J_p \dot{\theta}_p + \frac{1}{2}(k_\theta + k_\psi)\theta_p + \frac{1}{2}(k_\theta - k_\psi)\theta_r = -\Omega^2(J_p - J_T)\theta \]  

(51)

\[ J_T \ddot{\theta}_r + i\Omega J_p \dot{\theta}_r + \frac{1}{2}(k_\theta + k_\psi)\theta_r + \frac{1}{2}(k_\theta - k_\psi)\theta_p = 0 \]  

(52)

The equations 51 and 52 can be used for example where the rotor support is the same \( k_\theta = k_\psi = k \). The natural frequencies are the homogeneous solution of the equations 51 and 52. The solution has an exponential form \( e^{i\omega t} \).

\[
\begin{bmatrix}
(k - \omega^2 J_T + \alpha \Omega J_p) & 0 \\
0 & (k - \omega^2 J_T - \alpha \Omega J_p)
\end{bmatrix}
\begin{bmatrix}
\theta_p \\
\theta_r
\end{bmatrix}
= 0
\]  

(53)

Where one mode has progressive motion (forward whirling) and another one has retrograde motion (backward whirling) and the natural frequencies are when one of either \( \theta_p \) or \( \theta_r \) is equal to zero. The frequencies can be calculated from equation 53 and the solutions are shown in equations 54-57.

Mode with progressive motion \((\theta_p \neq 0, \theta_r = 0)\)

\[ k - \omega^2 J_T + \alpha \Omega J_p = 0 \]  

(54)

\[ \omega = \frac{\Omega J_p}{2J_T} \pm \sqrt{\frac{k}{J_T} + \left( \frac{\Omega J_p}{2J_T} \right)^2} \rightarrow \frac{\Omega J_p}{J_T} - \frac{k}{J_T} \quad : \quad \Omega \rightarrow \infty \]  

(55)

Mode with retrograde motion \((\theta_r \neq 0, \theta_p = 0)\)

\[ k - \omega^2 J_T - \alpha \Omega J_p = 0 \]  

(56)

\[ \omega = -\frac{\Omega J_p}{2J_T} \pm \sqrt{\frac{k}{J_T} + \left( \frac{\Omega J_p}{2J_T} \right)^2} \rightarrow -\frac{\Omega J_p}{J_T} + \frac{k}{J_T} \quad : \quad \Omega \rightarrow \infty \]  

(57)

The result is shown in Figure 31 below and this the reason why forward whirling and backward whirling are separate from each other when the disk has a conical motion.
Figure 31: Showing the result of the retrograde and progressive motion
A.2 Modal analysis methods in Ansys

The other six methods to do a modal analysis in Ansys are explained below. To choose the right method is important to get the right natural frequencies and the shortest calculation time. Table 8 is showing the difference between the methods. More information of almost all the solvers that are used in the different methods can be found on www.wikipedia.org.

A.2.1 Reduced

The matrices are reduced when the reduced method is used. The method is faster than Subspace, see A.2.2, because of the reduced matrices. Reduced method is using the HBI algorithm (Householder-Bisection-Inverse iteration) to get the eigenvalues and eigenvectors. It works in such a way that it takes out a small set of degrees of freedom called Master DOF. By using the Master DOF you get an exact stiffness matrix but an approximate mass matrix. Thus, the results are depends on how well mass matrix is approximated [11].

A.2.2 Subspace

The method is using the subspace iteration technique that uses the generalized Jacobi iteration algorithm. The method is very exactly because it using the full mass and stiffness matrices and that this is the reason why it is slower than Block Lanczos. Subspace should be used where high accuracy is required or the selecting of Master DOF is not possible [11].

A.2.3 Block Lanczos

Block Lanczos should be used for large symmetric eigenvalue problems. It is using the same solver as Subspace but it is converging faster. The Block Lanczos is using Lanczos algorithm where the iteration is done by a block of vectors. The method is almost as accurate as the Subspace method and the Block Lanczos is faster than Subspace. The method is very good at searching after eigenfrequencies in an interval of frequencies. The convergence rate for finding eigenfrequencies in the low range is almost the same for finding in the middle or higher range of frequency [11].

A.2.4 PCG Lanczos

PCG Lanczos method is used for very large symmetric problem with DOF bigger than 500 000. The method is using Lanczos algorithm which is combined with the PCG iterative solver. This way makes it much faster than Subspace and Block Lanczos for large models which have well shaped 3D solid elements and when only a few of the lowest modes is requested. If the elements are poorly shaped or too many modes are requested, the solution time will be long. The method is not recommended for eigenvalues that are fare from zero [11].

A.2.5 Unsymmetric

Unsymmetric method is used for unsymmetric problems where the matrices are not symmetric. The method is using the full stiffness and mass matrices as Subspace method but the matrices can be unsymmetric. The method can calculate complex eigenvalues and eigenvectors. The real part of the eigenvalue is the natural frequency and the imaginary part is deciding if the system is stable or not. A negative imaginary
part means that the system is stable and a positive means that it is not. This is different compared with the Damped and QR Damped method [11].

A.2.6 Damped

Damped method is used for analysis where the damping cannot be ignored like for a bearing system on a rotor. The method is using the full mass, stiffness and damping matrix and they can be unsymmetrical. Lanczos algorithm is used to calculate complex eigenvalues and eigenvectors. The imaginary part of the eigenvalue is the frequency of the system and the real part is saying if the system is stable. If the real part is positive when the system is unstable and negative means that the system is stable [11].

Table 8: Comparing the different modal extracting tools in Ansys [11]

<table>
<thead>
<tr>
<th>Procedure</th>
<th>Where to use it</th>
<th>Matrices which being used</th>
</tr>
</thead>
<tbody>
<tr>
<td>Subspace</td>
<td>Symmetric</td>
<td>K,M</td>
</tr>
<tr>
<td>Block Lanczos</td>
<td>Symmetric</td>
<td>K,M</td>
</tr>
<tr>
<td>PCG Lanczos</td>
<td>Symmetric (Buckling is not working)</td>
<td>K,M</td>
</tr>
<tr>
<td>Unsymmetric</td>
<td>Asymmetric matrices</td>
<td>K*,M*</td>
</tr>
<tr>
<td>Damped</td>
<td>Symmetric and asymmetric damped system</td>
<td>K*,M*,C*</td>
</tr>
<tr>
<td>QR Damped</td>
<td>Symmetric and asymmetric damped system</td>
<td>K*,M*,C*</td>
</tr>
<tr>
<td>Reduced</td>
<td>Any (Buckling is not recommended)</td>
<td>K,M</td>
</tr>
</tbody>
</table>

K = stiffness matrix, M = mass matrix, C = damping matrix
* = the matrix can be unsymmetrical.
A.3 Damping in Ansys

There are many possibilities to include damping into the model in Ansys but it is important to do it right and to declare the right damping constants. For the model for this thesis work only the element damping was used but it is good to know the rest of damping constants to be able to use them in the future. Not all of damping constants can be used for all the harmonic response analyses. For a Full method analysis, the damping which is allow is mass matrix multiplier \( \text{alphad} \), the stiffness matrix multiplier \( \text{Betad} \), constant damping ratio \( \text{DMPRAT} \), stiffness matrix material \( \text{MP,DAMP} \) and constant stiffness matrix coefficient for a material \( \text{MP,DMPR} \).

The damping which is allowed for Mode Superposition method is almost the same as for the Full method except in MSUP is also allowing modal damping ratio \( \text{MDAMP} \). The damping matrix is constructed from Equation 58 in Ansys [11].

\[
[C] = \alpha[M] + (\beta + \beta_v)[K] + \sum_{j=1}^{N_m} \left( \beta_j^m + \frac{2}{\Omega} \beta_j^\xi \right)[K_j] + \sum_{k=1}^{N} [C_k] + [C_{\xi}]
\]  

(58)

Where:

- \( \alpha \) = mass matrix multiplier
- \([C]\) = damping matrix
- \([M]\) = mass matrix
- \(\beta\) = stiffness matrix multiplier
- \(\beta_v\) = variable stiffness matrix multiplier
- \([K]\) = stiffness matrix
- \(N_m\) = number of materials with DAMP or DMPR input
- \(\beta_j^m\) = stiffness matrix multiplier for material j (input DAMP)
- \(\beta_j^\xi\) = constant stiffness matrix coefficient for material j (input DMPR)
- \(\Omega_{ex}\) = excitation rotational speed
- \([K_j]\) = stiffness matrix for material j
- \(N_e\) = number of element with damping
- \([C_k]\) = element damping matrix
- \([C_{\xi}]\) = frequency-dependent damping matrix

### A.3.1 Mass matrix and stiffness matrix multiplier

\(\text{alphad}\) is the command for mass matrix multiplier and \(\text{Betad}\) is the command for stiffness matrix multiplier in Ansys. Both \(\alpha\) and \(\beta\) are dependent of the energy dissipation of structure. The damping is viscous and the formula for the damping in Equation 59.

\[
[C] = \alpha[M] + \beta[K]
\]  

(59)

Both \(\text{alphad}\) and \(\text{betad}\) can be used for analysis with both the Full and the Mode Superposition method. If also the material coefficient DAMP under material data is valued, it will be add to \(\beta\) [11].
A.3.2 The frequency-dependent damping matrix

The frequency-dependent damping matrix $C_{\xi}$ is defined from the damping ratio $\xi^d$ which is the damping ratio for the mode shape. The equation from which $C_{\xi}$ is calculated from Equation 60 and 61.

\[ \{\Phi_i^T\}^T \begin{bmatrix} C_{\xi} \end{bmatrix} \{\Phi_i\} = 2\xi^d_i \omega_i \]  
\[ \xi^d_i = \xi + \xi^m_i \]  

Where:
- $\{\Phi_i\}$ = shape of mode $i$
- $\xi$ = constant damping ratio (input DMPRAT)
- $\xi^d_i$ = damping ratio for mode $i$
- $\omega_i$ = eigenfrequency for mode $i$
- $\xi^m_i$ = modal damping ratio for mode shape $i$ (input MDAMP)

But normally $\xi^d_i$ is used direct and $C_{\xi}$ is not calculated [11].

A.3.2 Variable stiffness matrix multiplier

Variable stiffness matrix multiplier can be use for The Full and The Reduced method but not for The Mode Superposition method. It is used to give a constant damping ratio over the whole frequency range and it is calculated from the ratio between actual damping and critical damping. The damping is calculated from Equation 62 [11].

\[ \beta_c = \frac{2}{\Omega_{ex}} \xi \]  

Where:
- $\xi$ = constant damping ratio (input DMPRAT)
- $\Omega_{ex}$ = excitation rotational speed in the range between $\Omega_B$ and $\Omega_E$
- $\Omega_B = 2\pi f_B$
- $\Omega_E = 2\pi f_E$
- $f_B$ = beginning frequency (input FREQB,HARFRQ)
- $f_E$ = end frequency (input FREQE,HARFRQ)

A.3.3 Modal damping ratio

Modal damping ratio is declared for every mode and the damping ratio is different for every mode. The modal damping ratio puts in a table together with the mode. If constant damping ratio (DMPRAT) is defined the modal damping ratio is add together with it [11].
A.4 Commands in Ansys

This appendix is showing the most important commands when rotor dynamics calculations should be done. This is extra information for people who wants to do rotor dynamics calculations when the help manual for Ansys is not the easiest one to understand. All the information of commands is from the Help Topic in Ansys Classic [11].

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.

Pstress, on
Prestress is used when the structure is prestressed 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 prestressed should be account in the model. The command is saying to the program that the eigenvalues should be saved for all the substeps.

Plcamp,, 1, rpm
Plcamp is used in the postprocessing 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.

Prcomp,, 1, rpm
Prcomp 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 can not 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.

A.4.1 Commands to harmonic force response analysis
There are three different methods to calculate the response the Full, the Reduced and the Mode Superposition method in Ansys. Only the Full and Mode Superposition are working with the gyroscopic effect. Commands to do a harmonic analysis are following below.

Antype,harmonic sets the analysis type to harmonic response.

Hropt, method,maxmode, minmode, mcout
Method is solution method which should be used, FULL, REDUC or MSUP. Maxmode is the largest number mode which should be used during the calculation, only valid for MSUP. Minmode is the lowest number of mode which should be used, valid only for MSUP. Mcout is the modal coordinates output only valid for MSUP. Yes means that they write to a text file. No means that no output will be done.

Hrout,Reimky,clust,Mcout
Reimky is either on/off. ON means that the solver should print out the complex displacement as a real and an imaginary part. OFF means that it should print out the complex displacement as amplitude and phase angle. Clust is either ON/OFF. OFF means that it should have uniform length between every step. ON means it should take shorter step close to the modes. Valid only for MSUP. Mcout is either ON/OFF. Valid only for MSUP. OFF means that it should not print the contributions at each frequency. ON means that it should print out.
EQSLV, lab, toler, mult

Lab is which solver the method should use. The solver can be set to, Front, Sparse, Jcg and a few more. Front solver is a frontal direct solver. Sparse solver is for calculating complex values with unsymmetrical matrices.

Toler is the tolerance the program should use.

Nsubst, nstbtp, nsmx, nsbmn

Nstbtp is the number of substep the program should use for every load step.

Nsmx is the maximal number of substep the program should take if automatic time step is used.

Nsbmn is the minimum number of substep the program should take if automatic time step is used.

KBC, KEY

KEY is how the loads will act on the model.

0 means that the load will be ramped on every substep.

1 means that the load will be stepped on the model on the first substep.

HARFRQ, FREQB, FREQE

Defines the frequency range for the loads

FREQB is the start frequency

FREQE is the end frequency. How large every step is between these two frequencies are decided from the number of substeps.

A.4.2 Commands to modal analysis

There are seven different methods to solve a modal problem in Ansys. The methods are Block Lanczos, PCG Lanczos, Subspace, Reduced, Unsymmetric, Damped and QR Damped method. Which method the engineer should use depends of which kind of problem.

ANTYPE, MODAL

Define that a modal analysis should be done.

MODOPT, Method, NMODE, FREQB, FREQE, Cpxmode, Nrmkey

Method is what kind of method the program should use. There are seven kinds explained in section A.2.

NMODE is the number of modes the program should calculate.

FREQB is the start frequency.

FREQE is the end frequency.

Cpxmode is the choice if the program should calculate the complex eigenvalues. Either ON/OFF. ON means that the program should calculate complex eigenvalues. OFF means that it should not. Only valid for QR Damped method.

Nrmkey is either ON/OFF. It is how the mode shapes should be normalized to. ON means that is should be normalized to the unit. That means that the largest deflection of structure should be equal to one. OFF means that the mode shapes should be normalized to the mass matrix instead.

MXPAND, NMODE, FREQB, FREQE, Elcalc, SIGNIF

Defines how many modes should be expanded.

NMODE is the number of modes which should be expanded.
FREQB is the start frequency.
FREQE is the end frequency.
Elcalc is if the program should calculate the element reaction and reactions forces in the model during the modal analysis. Either ON or OFF.
SIGNIF defines threshold which tells the program to only expand modes over this threshold.

A.5 Batch files and Matlab code for rotor with overhung disk

To do verify Ansys against theoretical calculations with a simple model, in Figure 13, some batch files had to written and they are shown in A.5.1-A.5.6. Three Matlab codes are written to calculate natural frequencies and the response to compare it with the results from Ansys. They are shown in A.5.7-A.5.9.

A.5.1 Batch file for the modal analysis for Case three

The batch file is written for Ansys to calculate the real frequencies for Case three to be able to draw a Campbell diagram which will be compared with the Matlab code in A.5.7. The results can be seen in Figure 15.

/PREP7
! Defines the elements
ET,1,MESH200
ET,2,SOLID186
KEYOPT,1,1,7
KEYOPT,1,2,0
! Defines the material data
MAT,1,
MP,Ex,1,211e9
MP,dens,1,7850
MP,prxy,1,0.3
! Defines the 2D model
BLC4,0,0,1,0.015
BLC4,1,0,0.05,0.3
aadd,all
lsel,,loc,x,0
lesize,all,,1
allsel
lsel,,loc,y,0
lsel,a,loc,y,0.015
lsel,u,loc,x,1,1.05
lesize,all,,25
allsel
lsel,,loc,x,1,1.05
lsel,u,loc,x,1
lsel,u,loc,x,1.05
lesize,all,,2
allsel
lsel,,loc,x,1
lsel,a,loc,x,1.05
lesize,all,0.03
allsel
type,1
amesh,all
  ! Rotate the 2D model 360 degrees
type,2
EXTOPT,ESIZE,3,0,
EXTOPT,ACLEAR,1
EXTOPT,ATTR,1,1,2
VROTAT,all,,1,2,360,,
ACLEAR,all
cm,e_model,elem

FINISH
/SOLU
asel,,loc,x,0
DA,all,ALL,
allsel
eplo
coriolis,on,,on
pi=acos(-1)
antype,modal
modopt,qrdamp,5,0.1,,on,
! Solving the modal analysis and writes out the result to utdata2.txt
*do,i,1,51
  spn=(i-1)*14*2*pi/60
cmomega,e_model,spn,,,0,0,0,1,0,0
mxpand,5
solve

*enddo
finish
/post1
neigv=5
nint=51
speeddel=14
pi=acos(-1)
*dim,rew,table,nint,neigv+1
*dim,imw,table,nint,neigv+2
*do,i,1,nint
nrpm=(i-1)*speeddel
*set,rew(i,1),nrpm
*set,imw(i,1),nrpm
*set,imw(i,2),nrpm/60
*do,j,1,neigv
  set,i,j,0,0
*get,rew(i,j+1),active,,set,freq
A.5.2 Batch file for the response analysis for Case three

The batch file is written for Ansys to calculate the response for Case three to be able to draw a response which will be compared with the Matlab code in A.5.7. The result can be seen in Figure 16.

/PREP7
! Defining the elements
ET,1,MESH200
ET,2,SOLID186
KEYOPT,1,1,7
KEYOPT,1,2,0
! Defines the material data
MAT,1,
MP,Ex,1,211e9
MP,dens,1,7850
MP,prxy,1,0.3
! Defines the geometry
BLC4,0,0,1,0.015
BLC4,1,0,0.05,0.3
aadd,all
lsel,,loc,x,0
lesize,all,,,1
allsel
lsel,,loc,y,0
lsel,a,loc,y,0.015
lsel,u,loc,x,1,1.05
lesize,all,,,25
allsel
lsel,,loc,x,1,1.05
lsel,u,loc,x,1
lsel,u,loc,x,1.05
lesize,all,,,2
allsel
lsel,,loc,x,1
lsel,a,loc,x,1.05
lesize,all,0.03
allsel
type,1
amesh,all
! Rotate the 2D model 360 degrees

type,2
EXTOPT,ESIZE,3,0,
EXTOPT,ACLEAR,1
EXTOPT,ATTR,1,1,2
VROTAT,all, , , , ,1,2 ,360, ,
ACLEAR,all
cm,e_model,elem

FINISH
/SOLU
asel,,loc,x,0
DA,all,ALL,
allsel
epl
coriolis,on,,on
pi=acos(-1)
speed=141
*dim,test1,table,speed,5
! Solving the response analysis and writes out the major and minor axial to utdata3.txt
*do,i,1,speed
fdele,2,all
spn=(i-1)*5*2*pi/60
/solu
allsel
cmomega,e_model,spn
antype,harmonic
Hropt,full
hrout,on

f,2,fy,0.111*spn**2
f,2,fz,-0.111*spn**2
eqslv,front
nsubst,1
KBe,1
Harfrq,spn/(2*pi)
solve
finish
/post1
set,,,,0
*get,yc,node,2,u,y
*get,zc,node,2,u,z
set,,,,1
*get,ys,node,2,u,y
*get,zs,node,2,u,z
ys=-ys
zc=-zc
zs=-zs
tmp1=2*(ys*zs-zc*yc)
tmp2=ys**2+yc**2-zs**2-zc**2
*if, tmp2, eq, 0, then
*if, tmp1, gt, 0, then
beta = pi/4
*elseif, tmp1, lt, 0, then
beta = -pi/4
*else
beta = 0
*endif
*else
beta = atan2(tmp1, tmp2)/2
*endif

tmp1 = ys*cos(beta) + zs*sin(beta)
tmp2 = yc*cos(beta) - zc*sin(beta)
*if, tmp2, eq, 0, then
*if, tmp1, gt, 0, then
alfa = pi/2
*elseif, tmp1, lt, 0, then
alfa = -pi/2
*else
alfa = 0
*endif
*else
alfa = atan2(tmp1, tmp2)
*endif

major = sqrt((yc**2 + ys**2)*cos(beta)**2 + (zc**2 + zs**2)*sin(beta)**2 + 2*(ys*zs - yc*zc)*cos(beta)*sin(beta))
*if, abs(sin(alfa)), lt, 0.7071, then
minor = (-ys*sin(beta) + zs*cos(beta))/cos(alfa)
*else
minor = (yc*sin(beta) + zc*cos(beta))/sin(alfa)
*endif

test1(i,1) = spn*60/(2*pi)
test1(i,2) = major
test1(i,3) = minor
test1(i,4) = alfa*180/pi
test1(i,5) = beta*180/pi
finish
/solu
*enddo

*cfopen, utdata3, txt
*vwrite, test1(1,1), test1(1,2), test1(1,3), test1(1,4), test1(1,5)
(f8.2, f10.5, f10.5, f10.5, f10.5)
*cfclose
finish
A.5.3 Batch file for the modal analysis for Case four

This batch file is for Ansys to calculate the natural frequencies for Case four and the results are compared with the frequencies that are calculated in Matlab code in A.5.8. The results can be seen in Figure 17.

/PREP7

! Defining the elements
ET,1,MESH200
*
ET,2,SOLID186
KEYOPT,1,1,7
KEYOPT,1,2,0
! Defining the material data
MAT,1,
MP,Ex,1,211e9
MP,dens,1,7850
MP,prxy,1,0.3
! Defining the geometry
BLC4,0,0,1,0.015
BLC4,1,0,0.05,0.3
aadd,all
lsel,,loc,x,0
lesize,all,,1
allsel
lsel,,loc,y,0
lsel,a,loc,y,0.015
lsel,u,loc,x,1,1.05
lesize,all,,25
allsel
lsel,,loc,x,1,1.05
lsel,u,loc,x,1
lsel,u,loc,x,1.05
lesize,all,,2
allsel
lsel,,loc,x,1
lsel,a,loc,x,1.05
lesize,all,0.03
allsel
type,1
amesh,all
! Rotate the 2D model 360 degrees
type,2
EXTOPT,ESIZE,3,0,
EXTOPT,ACLEAR,1
EXTOPT,ATTR,1,1,2
VROTAT,all, , , , ,1,2 ,360, ,
ACLEAR,all
cm,e_model,elem
! Defines the combi214 element with the spring constants
n,10000,1.025,0,0
et,3,214
keyopt,3,2,1
r,3,0.3e6,0.6e6
type,3
real,3
e,4,10000
d,10000,all
FINISH
/SOLU
asel,,loc,x,0
DA,all,ALL,
allsel
eplot
coriolis,on,,on
pi=acos(-1)
antype,modal
modopt,qrdamp,5,0.1,,on,
! Solve the problem and write out the solution to a text file utdata2.txt
*do,i,1,51
  
  spn=(i-1)*14*2*pi/60
cmomega,e_model,spn,,0,0,0,1,0,0
mxpand,5
solve
  
*endo
d finish
/post1
  
neigv=5
nint=51
speeddel=14
pi=acos(-1)
*dim,rew,table,nint,neigv+1
*dim,imw,table,nint,neigv+2
*do,i,1,nint
  nrpm=(i-1)*speeddel
  *set,rew(i,1),nrpm
  *set,imw(i,1),nrpm
  *set,imw(i,2),nrpm/60
  *do,j,1,neigv
    *set,i,j,0,0
    *get,rew(i,j+1),active,,set,freq
    set,i,j,0,1
  *get,imw(i,j+2),active,,set,freq
*endo
*endo
*cfopen,utdata2.txt
ii=1
*vwrite,imw(ii,1),imw(ii,2),imw(ii,3),imw(ii,4),imw(ii,5),imw(ii,6),imw(ii,7)
(f7.0,f6.1,f8.2,f8.2,f8.2,f8.2,f8.2)
**A.5.4 Batch file for the response analysis for Case four**

This batch file is for Ansys to calculate the response for Case four and the results are compared with the Matlab code in A.5.8. The results can be seen in Figure 18.

/ PREP7

! Defining the elements

ET,1,MESH200

!*

ET,2,SOLID186
KEYOPT,1,1,7
KEYOPT,1,2,0

! Defining the material data

MAT,1,
MP,Ex,1,211e9
MP,dens,1,7850
MP,prxy,1,0.3

! Making the geometry

BLC4,0,0,0,0.015
BLC4,1,0,0.05,0.3

add,all

lesel,,loc,x,0
lesize,all,,1
allsel

lesel,,loc,y,0
lesel,a,loc,y,0.015
lesel,u,loc,x,1,1.05
lesize,all,,25
allsel

lesel,,loc,x,1,1.05
lesel,u,loc,x,1
lesel,u,loc,x,1.05
lesize,all,,2
allsel

lesel,,loc,x,1
lesel,a,loc,x,1.05
lesize,all,0.03
allsel
type,1
amesh,all

! Rotate the 2D geometry 360 degrees
type,2
EXTOPT,ESIZE,3,0,
EXTOPT,ACLEAR,1
EXTOPT,ATTR,1,1,2
VROTAT,all,,1,1,2,360,,
ACLEAR,all
cm,e_model,elem
n,10000,1.05,0,0
et,3,214
keyopt,3,2,1
r,3,3e5,6e5
type,3
real,3
c,2,10000
d,10000,all
FINISH
/SOLU
asel,,loc,x,0
DA,all,ALL,,
allsel
eplo
coriolis,on,,on
pi=acos(-1)
speed=71
! Solving the problem and writes out the major axial and minor axial to utdata3.txt
*dim,test1,table,speed,5
*do,i,1,speed
fdele,2,all
spn=(i-1)*10*2*pi/60
/solu
allsel
cmomega,e_model,spn
antype,harmonic
Hropt,full
hrout,on

f,2,fy,0.111*spn**2
f,2,fz,,0.111*spn**2
eqslv,front
nsubst,1
KBe,1
Harfrq,spn/(2*pi)
solve
finish
/post1
set,,,,0
*get,yc,node,2,u,y
*get,zc,node,2,u,z
set,,,,1
*get,ys,node,2,u,y
*get,zs,node,2,u,z
ys=-ys
zc=-zc
zs=-zs
tmp1=2*(ys*zs-zc*yc)
tmp2=ys**2+yc**2-zs**2-2-zc**2
*if, tmp2,eq,0,then
*if, tmp1,gt,0,then
beta=pi/4  
*elseif,tmp1,lt,0,then  
  beta=-pi/4  
*else  
*endif  
*else  
  beta=atan2(tmp1,tmp2)/2  
*endif  
  
tmp1=ys*cos(beta)+zs*sin(beta)  
tmp2=yc*cos(beta)-zc*sin(beta)  
*if, tmp2,eq,0,then  
  *if,tmp1,gt,0,then  
    alfa=pi/2  
  *elseif,tmp1,lt,0,then  
    alfa=-pi/2  
  *else  
  *endif  
*else  
  alfa=atan2(tmp1,tmp2)  
*endif  
  
major=sqrt((yc**2+ys**2)*cos(beta)**2+(zc**2+zs**2)*sin(beta)**2+2*(ys*zs-yc*zc)*cos(beta)*sin(beta))  
*if,abs(sin(alfa)),lt,0.7071,then  
  minor=(-ys*sin(beta)+zs*cos(beta))/cos(alfa)  
*else  
  minor=(yc*sin(beta)+zc*cos(beta))/sin(alfa)  
*endif  
  
test1(i,1)=spn*60/(2*pi)  
test1(i,2)=major  
test1(i,3)=minor  
test1(i,4)=alfa*180/pi  
test1(i,5)=beta*180/pi  
finish  
/solu  
*enddo  
  
*cfopen,utdata3,txt  
*vwrite,test1(1,1),test1(1,2),test1(1,3),test1(1,4),test1(1,5)  
(f8.2,f10.5,f10.5,f10.5,f10.5)  
*cfclose  
Finish  

A.5.5 Batch file for the modal analysis for Case five  
This batch file is written for Ansys to calculate the natural frequencies for Case five which will be compared with the Matlab code in A.5.9. The results can be seen in Figure 19.
/PREP7
!Defining the elements
ET,1,MESH200
!
ET,2,SOLID186
KEYOPT,1,1,7
KEYOPT,1,2,0
! Defining the material data
MAT,1,
MP,Ex,1,211e9
MP,dens,1,7850
MP,prxy,1,0.3
! Defining the geometry
BLC4,0,0,1,0.015
BLC4,1,0,0.05,0.3
aadd,all
lsel,,loc,x,0
lesize,all,,,1
allsel
lsel,,loc,y,,0
lsel,a,loc,y,0.015
lsel,u,loc,x,1,1.05
lesize,all,,,25
allsel
lsel,,loc,x,1,1.05
lsel,u,loc,x,1
lsel,u,loc,x,1.05
lesize,all,,,2
allsel
lsel,,loc,x,1
lsel,a,loc,x,1.05
lesize,all,0.03
allsel
type,1
amesh,all
! Rotate the 2D model 360 degrees
type,2
EXTOPT,ESIZE,3,0,
EXTOPT,ACLEAR,1
EXTOPT,ATTR,1,1,2
VROTAT,all,,1,2,1,360,,
ACLEAR,all
cm,e_model,elem
! Defines the combi214 element with the spring constants
n,10000,1.025,0,0
et,3,214
keyopt,3,2,1
r,3,0.3e6,0.6e6,,,500,250
type,3
real,3
\begin{verbatim}
e,4,10000
d,10000,all
FINISH
/SOLU
asel,,loc,x,0
DA,all,ALL,
allsel
eplo
coriolis,on,,on
pi=acos(-1)
antype,modal
modopt,qrdamp,5,0.1,,on,
! Solve the problem and write out the solution to a text file utdata2.txt
*do,i,1,51
  spn=(i-1)*14*2*pi/60
cmomega,e_model,spn,,,0,0,0,1,0,0
mxpand,5
solve
*enddo
finish
/post1
neigv=5
nint=51
speeddel=14
pi=acos(-1)
*dim,rew,table,nint,neigv+1
*dim,imw,table,nint,neigv+2
*do,i,1,nint
  nrpm=(i-1)*speeddel
  *set,rew(i,1),nrpm
  *set,imw(i,1),nrpm
  *set,imw(i,2),nrpm/60
  *do,j,1,neigv
    set,i,j,0,0
    *get,rew(i,j+1),active,,set,freq
    set,i,j,0,1
    *get,imw(i,j+2),active,,set,freq
  *enddo
  *enddo
*cfopen,utdata2.txt
ii=1
*vwrite,imw(ii,1),imw(ii,2),imw(ii,3),imw(ii,4),imw(ii,5),imw(ii,6),imw(ii,7)
(f7.0,f6.1,f8.2,f8.2,f8.2,f8.2,f8.2)
*cfclose
\end{verbatim}
A.5.6 Batch file for the response analysis for Case five

This batch file is written for Ansys to calculate the response of Case five and the response curve will be compared with the response curve from the Matlab code in A.5.9. The results can be seen in Figure 20.

/PREP7

! Defining the elements
ET,1,MESH200
!* ET,2,SOLID186
KEYOPT,1,1,7
KEYOPT,1,2,0
! Defining the material data
MAT,1,
MP,Ex,1,211e9
MP,dens,1,7850
MP,prxy,1,0.3
! Making the geometry
BLC4,0,0,1,0.015
BLC4,1,0,0.05,0.3
aadd,all
lsel,,loc,x,0
lesize,all,,,1
allsel
lsel,,loc,y,,0
lsel,a,loc,y,0.015
lsel,u,loc,x,1,1.05
lesize,all,,,25
allsel
lsel,,loc,x,1,1.05
lsel,u,loc,x,1
lsel,u,loc,x,1.05
lesize,all,,,2
allsel
lsel,,loc,x,1
lsel,a,loc,x,1.05
lesize,all,0.03
allsel
type,1
amesh,all
! Rotate the 2D geometry 360 degrees
type,2
EXTOPT,ESIZE,3,0,
EXTOPT,ACLEAR,1
EXTOPT,ATTR,1,1,2
VROTAT,all, , , , ,1,2 ,360, ,
ACLEAR,all
cm,e_model,elem
n,10000,1.05,0,0
et,3,214
keyopt,3,2,1
r,3,3e5,6e5,,500,250
type,3
real,3
e,2,10000
d,10000,all
FINISH
/SOLU
asel,,loc,x,0
DA,all,ALL, allsel
eplo
coriolis,on,,on
pi=acos(-1)
speed=71
! Solving the problem and writes out the major axial and minor axial to utdata3.txt
*dim,test1$table,speed,5
*do,i,1,speed
fdele,2,all
spn=(i-1)*10*2*pi/60
/solu
allsel
cmomega,e_model,spn
antype,harmonic
Hropt,full
hrout,on

f,2,fy,0.111*spn**2
f,2,fz,0.111*spn**2
eqslv,front
nsubst,1
KBc,1
Harfrq,spn/(2*pi)
solve
finish
/post1
set,,,,0
*get,yc,node,2,u,y
*get,zc,node,2,u,z
set,,,,1
*get,ys,node,2,u,y
*get,zs,node,2,u,z
ys=-ys
zc=-zc
zs=-zs
tmp1=2*(ys*zs-zc*yc)
tmp2=ys**2+yc**2-zs**2-zc**2
*if, tmp2,eq,0,then
*if, tmp1,gt,0,then
beta=pi/4
*elseif,tmp1,lt,0,then
beta=-pi/4
*else
beta=0
*endif
*else
beta=atan2(tmp1,tmp2)/2
*endif
tmp1=ys*cos(beta)+zs*sin(beta)
tmp2=yc*cos(beta)-zc*sin(beta)
*if,tmp2,eq,0,then
*if,tmp1,gt,0,then
alfa=pi/2
*elseif,tmp1,lt,0,then
alfa=-pi/2
*else
alfa=0
*endif
*else
alfa=atan2(tmp1,tmp2)
*endif
major=sqrt((yc**2+ys**2)*cos(beta)**2+(zc**2+zs**2)*sin(beta)**2+2*(ys*zs-
yc*zc)*cos(beta)*sin(beta))
*if,abs(sin(alfa)),lt,0.7071,then
minor=(-ys*sin(beta)+zs*cos(beta))/cos(alfa)
*else
minor=(yc*sin(beta)+zc*cos(beta))/sin(alfa)
*endif
test1(i,1)=spn*60/(2*pi)
test1(i,2)=major
test1(i,3)=minor
test1(i,4)=alfa*180/pi
test1(i,5)=beta*180/pi
finish
/solu
*enddo

*cfopen,utdata3.txt
*vwrite,test1(1,1),test1(1,2),test1(1,3),test1(1,4),test1(1,5)
(f8.2,f10.5,f10.5,f10.5,f10.5)
*cfclose
Finish

**Matlab file for Case three**
The Matlab code is written to calculate the natural frequencies and the response for
Case three which is compared with the results from the batch files in A.5.1-A.5.2. The
results can be seen in Figure 15 and 16.
clear all; close all;
L=1;
\( E = 211 \times 10^9 \); 
\( \text{dens} = 7850 \); 
\( d = 0.03 \); 
\( h = 0.05 \); 
\( D = 0.6 \); 
\( m = 110 \); 
\( e = 0.001 \); 
\( I = \pi d^4/64 \); 
\( J_d = mD^2/16 \); 
\( J_p = mD^2/8 \); 
\( k_{11} = 12E*I/L^3 \); 
\( k_{12} = 6E*I/L^2 \); 
\( k_{22} = 4E*I/L \); 
\( M = \begin{bmatrix} m & 0 & 0 & 0 \\ 0 & m & 0 & 0 \\ 0 & 0 & J_d & 0 \\ 0 & 0 & 0 & J_d \end{bmatrix} \); 
\( K = \begin{bmatrix} k_{11} & 0 & 0 & -k_{12} \\ 0 & k_{11} & k_{12} & 0 \\ 0 & k_{12} & k_{22} & 0 \\ -k_{12} & 0 & 0 & k_{22} \end{bmatrix} \); 
\( \text{vetx} = [] \); 
\( \text{vety} = [] \); 
\( \text{vet1} = [] \); 

% Natural frequency 

for \( a = 1:351 \) 
\[ \omega = (a-1) \times 2 \times 2 \times \pi / 60; \]
\[ \text{vetx} = [\text{vetx}; \omega]; \]
\[ G = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ \omega & J_p & 0 & 0 \end{bmatrix}; \]
\[ [V, \text{ee}] = \text{polyeig} (K, G, M); \]
\[ \text{eee} = \text{sortrows} (\text{ee}); \]
\[ \text{vet1} = [\text{vet1}; \text{eee}']; \]
if \( a = 1 \) 
\[ E(1) = \text{imag} (\text{eee}(3)); \]
\[ E(2) = \text{imag} (\text{eee}(4)); \]
\[ E(3) = \text{imag} (\text{eee}(7)); \]
\[ E(4) = \text{imag} (\text{eee}(8)); \]
else 
for \( j = 1:4 \) 
\[ E(j) = \text{imag} (\text{eee}(2j)); \]
end 
end 
\[ \text{vety} = [\text{vety}; E]; \]
end 
\[ \text{vetx} = \text{vetx} / (2 \times \pi); \]
\[ \text{vety} = \text{vety} \times 60 / (2 \times \pi); \]
\[ \text{MM} = \text{dlmread} ('\text{campbellsimp 23mars.txt}'); \]
\[ \text{AA} = \text{size} (\text{MM}); \]
figure(1); 
\[ \text{xlabel} ('\text{Rotational speed, [rpm]}'); \]
\[ \text{ylabel} ('\text{Natural Frequency, [Hz]}'); \]
\[ \text{plot} (\text{vetx}, \text{vety}(:, 1), '-'); \]
hold on 
\[ \text{plot} (\text{vetx}, \text{vety}(:, 2), '-'); \]
\[ \text{plot} (\text{vetx}, \text{vety}(:, 3), '-'); \]
\[ \text{plot} (\text{vetx}, \text{vety}(:, 4), '-'); \]
\[ \text{plot} (\text{vetx}, \text{vetx} / 60, '-'); \]
for \( i = 1: (\text{AA}(2) - 2) \) 
\[ \text{plot} (\text{MM}(:, 1) \text{MM}(:, i+2), '-'); \]
end 
hold off 
% Response analysis 
\[ \text{vetx1} = []; \]
\[ \text{vety1} = []; \]
\[ \text{vet} = []; \]
\[ F = 0; \]
for \( a = 1:401 \)
omega=(a-1)*1.75*2*pi/60;
vtx1=[vtx1;omega];
G=[0 0 0 0;0 0 0 0;0 0 omega*Jp;0 0 -omega*Jp 0];
F=m*e*omega^2;
Fx=[F 0 0 0];
Fy=[0 F 0 0];
FF=[Fx' Fy'];
AB=CC\FF;
q0=(AB(1)^2+AB(2)^2)^0.5;
vety1=[vety1;q0];
end

BB=[];
BB=dlmread('siemensresp 23 mars.txt');
figure(3)
plot(vtx1*60/(2*pi),vety1,'-');
hold on
plot(BB(:,1),BB(:,2),'--','color','green');
xlabel('Rotational speed, [rpm]');
ylabel('Radial displacement, [m]');
axis([0 700 0 0.004]);
hold off

A.5.8 Matlab file for Case four

The Matlab code is written to calculate the natural frequencies and the response for Case four which is compared with the results from the batch files in A.5.3-A.5.4. The results can be seen in Figure 17 and 18.

clear all; close all;
L=0.99;
E=211*10^9;
dens=7850;
d=0.03;
h=0.05;
D=0.6;
m=110;
e=0.001;
I=pi*d^4/64;
Jd=m*D^2/16;
Jp=m*D^2/8;
k11=12*E*I/L^3;
k12=6*E*I/L^2;
k22=4*E*I/L;
kx=0.3*10^6;
ky=0.6*10^6;
M=[m 0 0 0;0 m 0 0;0 0 Jd 0;0 0 0 Jd];
K=[k11+kx 0 -k12;0 k11+ky k12 0;0 k12 k22 0;-k12 0 0 k22];
vtx=[];
vety=[];
MM=[];

%Natural frequency
for k=1:51
    omega=(k-1)*14*2*pi/60;
vtx=[vtx;omega];
G=[0 0 0 0;0 0 0 0;0 0 omega*Jp;0 0 -omega*Jp 0];
[V,ee]=polyeig(K,G,M);
eee=sortrows(ee);
    for j=1:4;

- 27 -
E(j)=imag(eee(2*j));
end
vety=[vety;E];
end

MM=dlmread('campbell 23 mars.txt');
AA=size(MM);
vety=vety/(2*pi);
vetx=vetx*60/(2*pi);
figure(1);
xlabel('Rotational speed, [rpm]');
ylabel('Natural Frequency, [Hz]');
plot(vetx,vety(:,1),'-');
hold on
plot(vetx,vety(:,2),'-');
plot(vetx,vety(:,3),'-');
plot(vetx,vety(:,4),'-');
plot(vetx,vetx/60,'-');
for i=1:(AA(2)-2);
    plot(MM(:,1),MM(:,i+2),'--');
end
hold off
%Response analysis
vetx1=[];
vetyx1=[];
vety1y=[];
vetalfa=[];
F=0;
vety1=[];
for a=1:101
    omega=(a-1)*7*2*pi/60;
    vetx1=[vetx1;omega];
    G=[0 0 0 0;0 0 0 0;0 0 0 omega*Jp;0 0 -omega*Jp 0];
    F=m*e*omega^2;
    Fx=[F 0 0 0];
    Fy=[0 F 0 0];
    CC=[K-omega^2*M G*omega;-omega*G K-omega^2*M];
    FF=[Fx';Fy'];
    AB=CC\FF;
    xc=AB(1);
    yc=-AB(2);
    ys=AB(6);
    xs=AB(5);
    tmp1=2*(xs*ys-xc*yc);
    tmp2=xs^2+xc^2-ys^2-yc^2;
    if tmp2==0
        beta=pi/4;
    elseif tmp1<0
        beta=-pi/4;
    else beta=0;
    end
else
    beta=atan2(tmp1,tmp2)/2;
end
tmpl1=xs*cos(beta)+ys*sin(beta);
tmpl2=xc*cos(beta)-yc*sin(beta);
if tmpl2<0
    if tmpl1<0
        alfa=pi/2;
    elseif tmpl1>0
        beta=pi/4;
    elseif tmpl1<0
        beta=-pi/4;
    else beta=0;
    end
else
    beta=atan2(tmpl1,tmpl2)/2;
end
alfa=-pi/2;
else alfa=0;
end
else
alfa=atan2(tmp1,tmp2);
end
major=((xc^2+xs^2)*(cos(beta))^2+(yc^2+ys^2)*(sin(beta))^2+2*(xs*ys-xc*yc)*sin(beta)*cos(beta))^0.5;
if abs(sin(alfa))<0.7071
    minor=(-xs*sin(beta)+ys*cos(beta))/cos(alfa);
else
    minor=(xc*sin(beta)+yc*cos(beta))/sin(alfa);
end
vetbeta=[vetbeta;beta];
vetalfa=[vetalfa;alfa];
verty1x=[verty1x;major];
verty1y=[verty1y;minor];
vetab=[vetab;AB'];
end
BB=[];
BB=dlmread('Fjader 141 steg 23mars.txt');
figure(3)
plot(vetx1*60/(2*pi),verty1x,'-','color','red');
xlabel('Rotational speed,[rpm]');
ylabel('Radial displacement, [m]');
hold on
plot(vetx1*60/(2*pi),verty1y,'-');
plot(BB(:,1),BB(:,2),'--','color','green');
plot(BB(:,1),BB(:,3),'--','color','black');
axis([0 700 -0.005 0.025]);
hold off
figure(4)
plot(vetx1*60/(2*pi),vetbeta*180/pi,'-','color','red');
hold on
plot(vetx1*60/(2*pi),vetalfa*180/pi,'-');
plot(BB(:,1),BB(:,4),'--','color','green');
plot(BB(:,1),BB(:,5),'--','color','black');
hold off

A.5.9 Matlab file for Case five

The Matlab code is written to calculate the natural frequencies and the response for Case five which is compared with the results from the batch files in A.5.5-A.5.6. The results can be seen in Figure 19 and 20.

clear all; close all;
L=0.99;
E=211*10^9;
dens=7850;
d=0.03;
h=0.05;
D=0.6;
m=110;
e=0.001;
I=pi*d^4/64;
Jd=m*D^2/16;
Jp=m*D^2/8;
k11=12*E*I/L^3;
k12=6*E*I/L^2;
$k_{22} = 4 * E * I / L$
$K_{xx} = 3 \times 10^5$
$K_{yy} = 6 \times 10^5$
$C_{xx} = 500$
$C_{yy} = 250$

$M = \begin{bmatrix} m & 0 & 0 & 0 \\ 0 & m & 0 & 0 \\ 0 & 0 & J_d & 0 \\ 0 & 0 & 0 & J_d \end{bmatrix}$
$K = \begin{bmatrix} K_{11} + K_{xx} & 0 & 0 & -K_{12} \\ 0 & K_{11} + K_{yy} & K_{12} & 0 \\ 0 & K_{12} & K_{22} & 0 \\ -K_{12} & 0 & 0 & K_{22} \end{bmatrix}$
$v_{ex} = []$
$v_{ey} = []$
$v_{ey1} = []$

%% Natural frequency
for $k = 1:51$
    $\omega = (k-1) * 14 * 2 \pi / 60$
    $v_{ex} = [v_{ex}; \omega]$
    $G = \begin{bmatrix} c_x & 0 & 0 & 0 \\ 0 & c_y & 0 & 0 \\ 0 & 0 & 0 & \omega * J_p \\ 0 & 0 & -\omega * J_p & 0 \end{bmatrix}$
    $[V, \text{ee}] = \text{polyeig}(K, G, M)$
    $\text{ee} = \text{sortrows}(\text{ee})$
    for $j = 1:4$
        $E(j) = \text{imag}(\text{ee}(2*j))$
        $EE(j) = \text{real}(\text{ee}(2*j))$
    end
    $v_{ey1} = [v_{ey1}; EE]$
    $v_{ey} = [v_{ey}; E]$
end
$v_{ey} = v_{ey} / (2 \pi)$
$v_{ey1} = v_{ey1} / (2 \pi)$
$v_{ex} = v_{ex} / 60 / (2 \pi)$
$MM = \text{dlmread('campbell 23 mars.txt')}$
$AA = \text{size}(MM)$
figure(1);
xlabel('Rotational speed, [rpm]');
ylabel('Natural Frequency, [Hz]');
plot(vex, vey(:,1), '-');
hold on
plot(vex, vey(:,2), '-');
plot(vex, vey(:,3), '-');
plot(vex, vey(:,4), '-');
plot(vex, vex/60, '-');
for $i = 1:(AA(2)-2)$
    plot(MM(:,1), MM(:,i+2), '--');
end
hold off
figure(2);
xlabel('Rotational speed, [rpm]');
ylabel('Re(eigenvalue), [Hz]');
plot(vex, vey1(:,1), '-');
hold on
plot(vex, vey1(:,2), '-');
plot(vex, vey1(:,3), '-');
plot(vex, vey1(:,4), '-');
hold off

% Response analysis
$v_{ex1} = []$
$v_{eyx} = []$
$v_{eyy} = []$
$v_{eta} = []$
$F = 0$
$v_{etab} = []$
$v_{etab} = []$
for $a = 1:201$
    $\omega = (a-1) * 3.5 * 2 \pi / 60$
vetx1=[vetx1;omega];
G=[cx 0 0 0;0 cy 0 0 0;0 0 0 omega*Jp;0 0 -omega*Jp 0];
F=me*omega^2;
Fx=[F 0 0 0];
Fy=[0 F 0 0];
CC=[K-omega^2*M G*omega;-omega*G K-omega^2*M];
FF=[Fx';Fy'];
AB=CC\FF;
xc=AB(1);
yc=-AB(2);
ys=AB(6);
xs=AB(5);
tmp1=2*(xs*ys-xc*yc);
tmp2=xs^2+xc^2-ys^2-yc^2;
if tmp2==0
    if tmp1>0
        beta=pi/4;
    elseif tmp1<0
        beta=-pi/4;
    else
        beta=0;
    end
else
    beta=atan2(tmp1,tmp2)/2;
end

tmp1=xs*cos(beta)+ys*sin(beta);
tmp2=xc*cos(beta)-yc*sin(beta);
if tmp2==0
    if tmp1>0
        alfa=pi/2;
    elseif tmp1<0
        alfa=-pi/2;
    else
        alfa=0;
    end
else
    alfa=atan2(tmp1,tmp2);
end
major=((xc^2+xs^2)*(cos(beta))^2+(yc^2+ys^2)*(sin(beta))^2+2*(xs*ys-xc*yc)*sin(beta)*cos(beta))^0.5;
if abs(sin(alfa))<0.7071
    minor=(-xs*sin(beta)+ys*cos(beta))/cos(alfa);
else
    minor=(xc*sin(beta)+yc*cos(beta))/sin(alfa);
end
vetbeta=[vetbeta;beta];
vetafa=[vetafa;alfa];
vetylx=[vetylx;major];
vetyly=[vetyly;minor];
veta=[veta;AB'];
BB=[];
BB=dlmread('damping 23 mars resp.txt');
figure(3)
plot(vetx1*60/(2*pi),vetylx,'-','color','red');xlabel('Rotational speed, [rpm]');ylabel('Radial diplacement, [m]');hold on
plot(vetx1*60/(2*pi),vetyly,'-');plot(BB(:,1),BB(:,2),'--','color','green');plot(BB(:,1),BB(:,3),'--','color','black');axis([0 700 -0.002 0.016]);hold off