International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Impact Factor (2012): 3.358

Parametric Approach for Vibration Analysis of a Centrifuge

Manjunath Kallur[1], S. K. Ambekar[2]

[1]Departament of Mechanical engineering, KLS's Gogte Institute of Technology, VTU Belgaum, Karnataka, India

[2]Professor, Department of Mechanical engineering, KLS's Gogte Institute of Technology, VTU Belgaum, Karnataka, India

Abstract: ***Centrifuge is a device that spins at high speeds up to 2000 RPM or more. It is used in process industry like sugar plants, food processing industry and pharmaceutical industry. It is useful for separating the solid particles from liquid-solid mixtures. It is for example used in sugar industry for separating sugar crystals from molasses (concentrated sugar syrup). The present work aims at developing a program using ansys parametric design language for investigating the stresses and vibrations acting on the centrifuge of various sizes and speed. The modeling and analysis of the centrifuge was done by using ANSYS APDL 14.0 and the validation of the program is done by analytical method.***

Keywords: Centrifuge, Stress analysis, ANSYS APDL, Parametric approach.

1. Introduction

The centrifuge is a kind of separating machine, it is mainly used to separate the mixed liquid of solid and liquid (or liquid and liquid), so then can get the solid and liquid (or liquid and liquid) apart. The centrifuge works on the principle that, stewing one mixed liquid within different liquids densities, the mixed one will appear natural stratification phenomenon. The solid part will settle to the bottom, and there will form clear liquid part in the upper level. Some mixed liquids are easy to be stratification, such as mud water; on the other hand, some of them can be stratification in quite a long time and not very clear. This kind of stratification depends on the gravity of earth. To meet the needs of industrial production, people require separating some mixed liquids much more and faster, so that the centrifuges are used. The centrifuges revolve at high speeds, and form powerful centrifugal force, usually the centrifugal separating factor is hundreds, thousands, and even ten thousands time than the gravity. Though the separating speed is fast, according to the different properties of different materials, and production rates many centrifuges with different size and specifications are being used in industry.

Generally, the speed of centrifuges used to separate solid and liquid is below 2000 r/min; to separate mixed liquids within more tiny particles and more little density differences, users need centrifuges with speed between 8000 and 30000.

Generally speaking, the centrifuge is a centrifugal separation equipment that the bowl driven by motor and revolve in a high speed, to separate the liquid with smaller density from material or carry out the settling, stratification, and separation. Simple centrifuges are braked by hand brake, operated by manual loading and unloading, but now the popular operation of centrifuge is by automatic loading, unloading by automatic scraper or lifting basket with frequency controlling technology. Because of the importance and broad applicability of centrifuges, and the technology developments are fast too, the centrifuges manufacturers are all working hard to manufacture more centrifuges with intelligent controlling and more convenient operation, to sell them to the domestic and international markets.

2. Working Principle

The centrifuge works by revolving its contents around in a circle and uses their inertia to separate them. The various particles present in the centrifuge will have different densities and other characteristics which affect their path as they start revolving around the center of the centrifuge. The inertia of the each particle tends to go them straight, while the centrifuge forces them to bend inward. The forces causing this inward bending have to be conveyed from the centrifuge through its contents and there is a tendency for the denser particles in the centrifuge to travel straighter than the less dense particles. As a result, the denser particles are found near the outside of the circular path while the less dense ones are found near the center of that path.

3. Problem Definition

The below figure 1 shows the cut section of centrifuge structure used in sugarcane industry. The centrifuge typically consists of two bucket shaped cylindrical stainless steel vessels that are mounted on stainless steel flanges. The buckets and flanges are spun at a high speed by a rotating shaft that is mounted in antifriction bearings.

These members are fabricated together by screwing with appropriate number of correct size screws. The high-speed centrifugal action used for the separation of massecuite into sugar crystals and molasses, and is done in revolving machines called centrifuges. A centrifuge structure has a cylindrical bucket suspended on a spindle/shaft, with perforated sides lined by wire cloth, and inside which are metal sheets containing 400 to 600 perforations per square inch. The bucket revolves at speeds from 1,000 to 1,800 RPM. The mother liquid/molasses passes through the lining (due to the centrifugal force exerted). The raw sugar crystals are retained in the centrifuge bucket, Because of the perforated linings on the bucket. The final molasses (blackstrap molasses) containing waste sucrose, water, ash, reducing sugars, and organic non-sugars, is sent to the large storage tanks at the bottom.

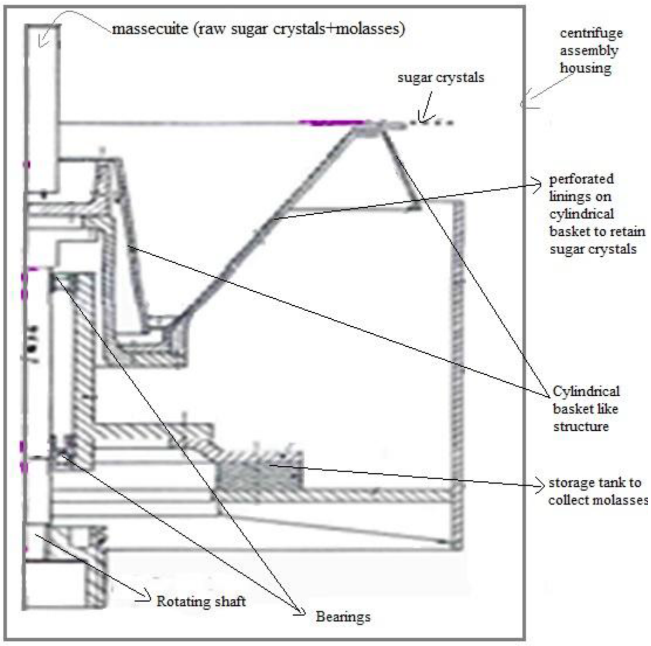

Figure 1: Cut section of centrifuge assembly used in sugar plants (2D drawing)

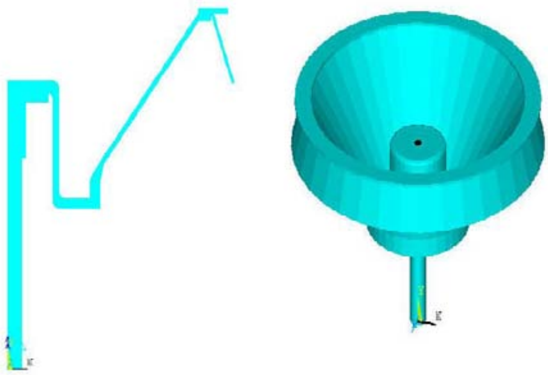

Figure 2: 3D Model of centrifuge

During the use, centrifugal forces develop in the rotating components of the centrifuge. Vibrations and stress as such develop in the spinning members. The vibrations can resonate the structures. The stresses can induce cracks in the members which can lead to total failure of the machine. The thickness of the members is to be decided based on the induced stress.

4. Objective of the Present Work

Investigation into the vibrations and computation of the stress fields is difficult by analytical methods. Recourse to FEM can help in arriving at the values. But, in view of various sizes of the centrifuges and speeds, it is worth while going through a parametric program approach. The program interactively prompts the designer to input some data of the machine. The program then generates the model and self drives the FEM procedure for predicting deflections, stresses and vibrations in the structure. This helps the designer to arrive at an optimum design in a short time. The program can be used for any size or speed of the machine.

5. Developing the APDL Program

ANSYS Parametric Design Language (APDL) is a design language, used to create the model and automate tasks with the help of parameters. A sequence of programming commands can be recorded in a macro file and these commands enables the user to build the customized ANSYS command that will execute all of the commands required for a particular analysis. APDL is the sophisticated language which is used in design optimization and adaptive meshing. Modal analysis using FEA packages has proven to be an effective tool in determining frequency of such a system.

The ANSYS APDL program has been developed to carry out the static and modal analysis of the structure. The program is written to simplify the whole analysis, as whole process is very cumbersome to analyze the structure. The program reduces overall time required for the analysis to be carried out and can be used to analyze the various structures of different size and speed more or less shape will be same. The developed ANSYS APDL program prompts the user to input dimensions, Boundary conditions and other parameter of the structure and the program is written in such a way that element type, material properties, meshing, boundary conditions etc, are automatically applied as per user inputted data and analysis is also done automatically to get the stresses and vibrations.

5.1 Finite Element Modeling

The finite element software ANSYS MECHANICAL APDL (version 14.0) is used for the analysis of centrifuge structure shown figure 2. The first step in the analysis is the 2D study of the structure. At first the element type required to model the structure needs to be chosen. A standard PLANE 183 element is used to model the centrifuge structure; PLANE183 is a higher order 2-D, 8-node or 6-node element. PLANE183 has quadratic pure displacement behavior and is well suited to modeling irregular meshes (such as those produced by various CAD/CAM systems).

The plane183 element is described by 8 nodes or 6 node which are having two degrees of freedom at each node i.e. Translations in the nodal x and y directions.

This element has plasticity, Stress stiffening, hyper-elasticity, creep, large deflections, and the large strain capabilities. The plane 183 element supports the axisymmetric loading conditions (The plane 183 element is chosen because our case is also axisymmetric loading). The axisymmetric model was created by drawing the half cross-section and then partitioning in different regions to facilitate material assignment (The three dimensional model was created by first sketching a half cross-section and then revolving it to 360 degrees, as shown in figure 2)

Table 1: Plane183 element description

Element name	PLANE183
Nodes	I, J, K, L, M, N, O, P
Degrees of Freedom	UX, UY
Real Constants	None
Material properties	EX, EY, EZ, PRXY, PRYZ, PRXZ, ALPX , DENS, BETD
Special Features	Elasticity, Element Technology Auto selects, Hyper elasticity, Large Deflection, Large Strain, Linear Perturbation, Nonlinear Stabilization, and Plasticity with User

5.2 Material Properties

The material used in this analysis is Stainless Steel (SS304), is assumed to be Linear-Elastic-Isotropic following the Von-Misses yield criterion. The elastic properties are the modulus of elasticity, E =210GPa, Poisson's ratio, υ = 0.3, Density, ρ = 7.85 e-6 kg/mm3 and tensile yield stress, σyield = 600~700 MPa.

5.3 Meshing

Meshing is an integral part of the analysis process. The mesh influences the accuracy, convergence and speed of the solution. The model was discretized using solid plane 183 element type. The figure 3 shows the meshing of centrifuge.

Figure 3: Finite Element Mesh of Centrifuge

The mesh is generated using free area mesh as it has no restrictions in terms of element shapes, has no specified pattern applied to it, easy to create geometry and meshing is fast as shown in figure 3. The nodes, elements, areas, properties of material, boundary conditions and other physical system defining parameters that constitute the model are created exclusively by using APDL commands such as ET, MAT, EX, PRXY, DENS, UX, UY, and DK.

5.4 Boundary Conditions

The finite element model of the centrifuge structure with applied boundary conditions is as shown in figure 4. Here the two antifriction bearings are mounted on the shaft at their respected locations (Bearing1 is constrained only at x direction, while bearing 2 is constrained at both x & y direction). The angular velocity ω in rpm is applied to axis of rotation y-axis.

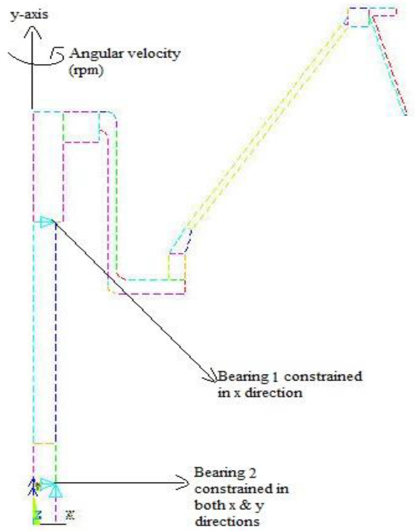

Figure 4: Boundary conditions applied

5.5 Structural Analysis

Structural analysis is the most common application of the finite element method; used to analyze the mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts and tools. The types of structural analyses are.

1) Static Analysis: Static analysis determines the effects of steady loading conditions on a body or structure, while ignores the inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes).
2) Modal Analysis: Modal analysis is carried out to find the vibrational characteristics (natural frequencies and their mode shapes) of a structure during the design stage itself. The goal of modal analysis in structural mechanics is to determine the natural mode shapes and frequencies of an object or structure during free vibration.

6. Results and Discussion

In the present study, Analysis is carried out to find maximum stress and deflections. The feasibility of design is considered based on structural design requirements. An APDL code is used to check the structural safety of the problem. By the post-processor of ANSYS, the nodal plot of von-mises stress and the nodal plot of resultant displacement in case of 2000rpm were plotted.

6.1 Displacement Results

The below figure 5 shows the deformation shape of the centrifuge structure, which is maximum at the outermost part of the structure because the structure is slender towards the outer bucket. And there is no deformation at the shaft because the shaft structure is very stiff as compared to the buckets. The value of maximum deformation is 0.008117mm.

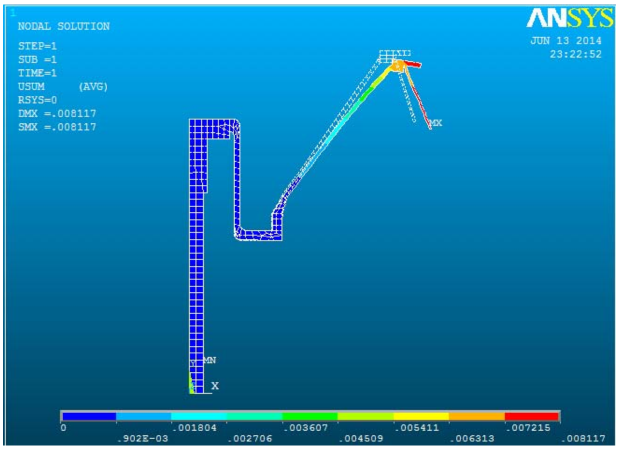

Figure 5: Deformation shape

6.2 Stresses acting on the Structure

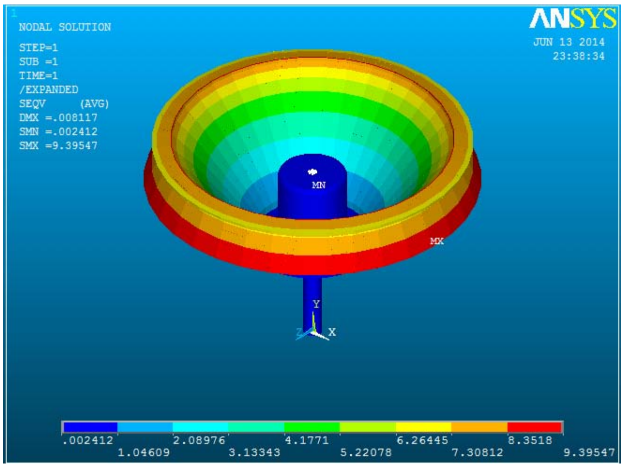

Figure 6: Von-Mises Stress

The figure 6 shows the von-mises stress acting on the structure, a spinning member experiences centrifugal stresses that results in stresses throughout the part and the von-mises stresses due to centrifugal forces are shown. Here the maximum stress is acting at the outer bucket of the structure because the structure is very slender at that part. The max stress value is 9.39547mpa which is much smaller than yield strength and tensile strength of the material, Hence the structure is safe.

6.3 Natural Frequencies and Mode Shapes

The centrifuge structure of stainless steel material is analyzed in the modal analysis to get the natural frequencies and their mode shapes. Table 2 shows frequencies at which the model resonates.

Table 2: Modal Analysis of Centrifuge

Set	Frequency (Hz)	Load Step	Sub Step	Cumulative
1	904.89	1	1	1
2	2995.1	1	2	2
3	3939.1	1	3	3
4	4943.3	1	4	4

The centrifuge operates at 2000rpm, i.e. 33.33 Hz. Since all the frequencies of different modes are much far than the working frequency. There is no resonance, hence the structure is considered to be safe. The below figures 7 to 10 shows the natural frequencies and their mode shapes

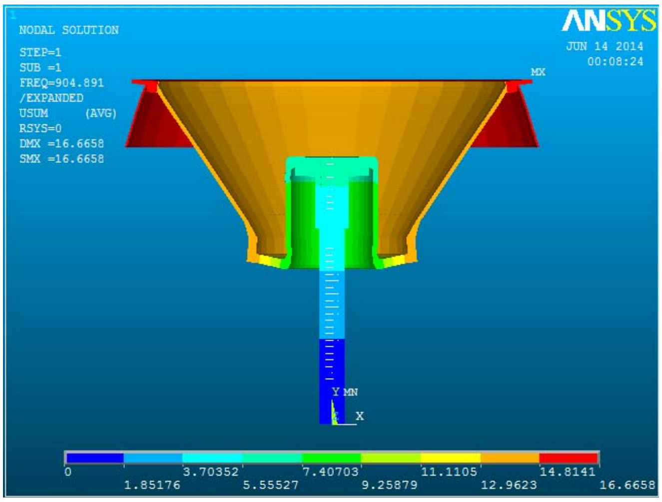

Figure 7: Mode shape 1 at frequency 904.891 Hz.

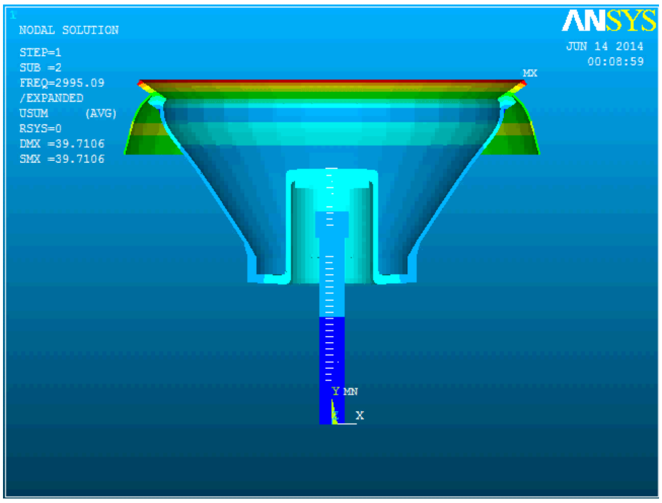

Figure 8: Mode shape 2 at frequency 2995.1 Hz.

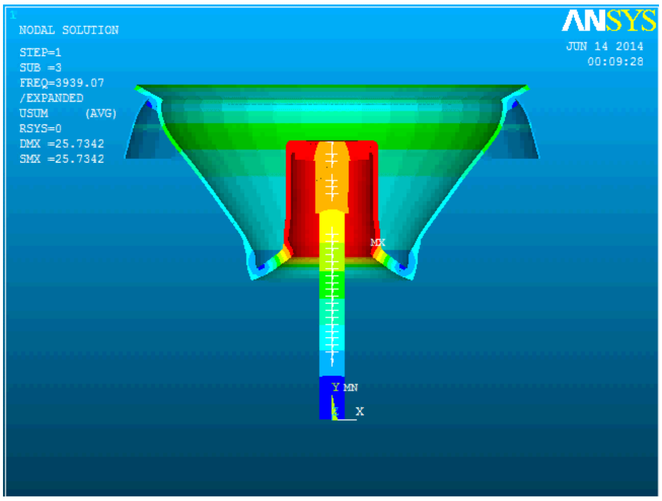

Figure 9: Mode shape 3 at frequency 3939.07 Hz.

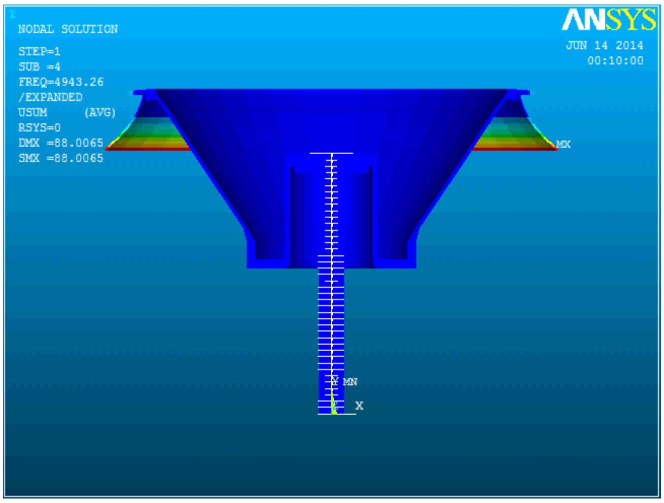

Figure 10: Mode shape 4 at frequency 4943.26 Hz.

6.4 Comparing the effect of stress & displacement on speed

To study the effect of stress and displacement (How stress & displacement varies with speed) on speed, the analysis is made keeping the same dimensions and varying the speed. Table 3 shows the effect of stress & displacement on speed

Table 3: Effect of stress and displacement on speed

Speed (rpm)	Displacement (mm)	Von-Mises Stress (Mpa)
1000	2.64961	278.814
1200	3.89226	409.375
1500	5.96161	627.332
2000	10.8036	1136.85

By comparing the values of stress and displacement w.r.t. speed, we can conclude that as the speed increases, displacement and stress also increases.

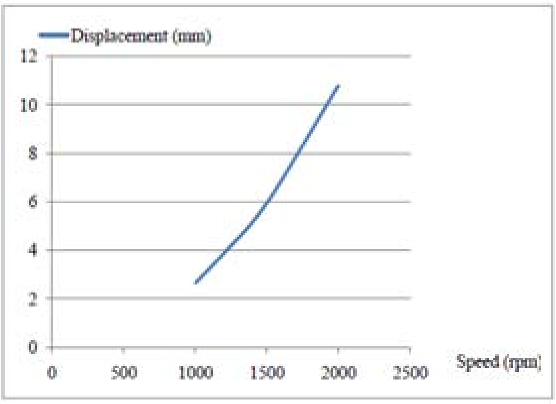

Figure 11: Graph of speed vs displacement

The figure 11 and 12 shows the relationship between speed, displacement and von-mises stress. With the help of above graphs one can study the limiting speed of the structure to prevent failure.

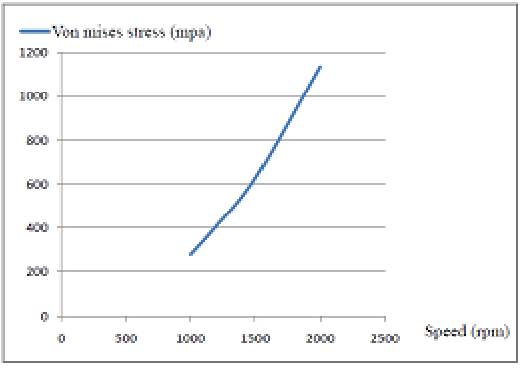

Figure 12: Graph of speed vs von-mises stress

Hence the written APDL program for the centrifuge structure perfectly works for any size and speed of the machine. The program can be used by anyone (even the person who doesn't have knowledge of Ansys). It also helps the designer to arrive at optimum design in a short time. It also reduces the time required for the analysis.

7. Analytical Validation of Program

In order to validate Ansys tool, the outer part of the bucket of centrifuge structure is considered as the thick cylinder (as the stress is maximum at the outer bucket) and hoop stress or circumferential stress is calculated. The main criteria for failure chosen are maximum strain energy criterion or von-mises failure criteria. It says that the material will fail when the equivalent stress exceeds the yield point limit.

Analytical calculation of hoop stress:

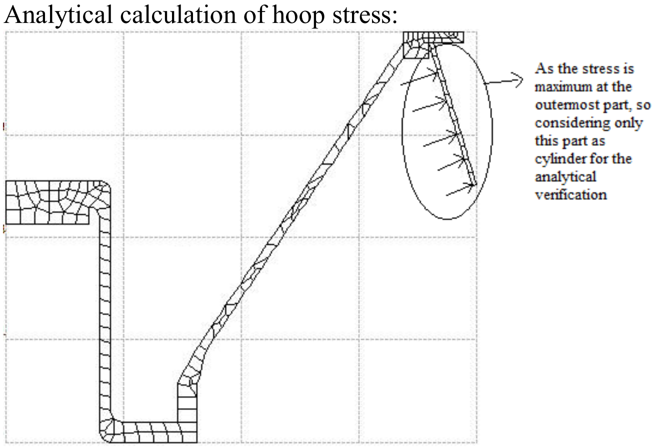

Figure 13: Cut section of centrifuge

The hoop stress or circumferential stress (1st principal stress) is given by $\sigma_1 = pd/2t$

Where p = internal pressure, d = diameter, t = thickness

p = F/A

Where F = centrifugal force, A = Area

$F = m\omega^2 r$

Where m = mass, w = angular rotation, r = radius

m = Volume (Length*Breadth*2πr) * Density

$m = 73562.27*7.85*10^{-6}$

m = 0.5774 kg

ω = 2πN/60

ω = 2*3.142*2000/60

ω = 209.439 rad/s

r = 150.1 mm

$F = 0.5774*10^{-3}*(209.439)^2*150.1$
$F = 3801.672$ N
$A = 2\pi r*L$
$A = 2*3.142*150.1*52$
$A = 49041.517$ mm^2
$p = F/A = 0.77519$

Now substituting above equations in hoop stress formula
$\sigma_1 = (0.077519*300.2)/(2*1.5)$
$\sigma 1 = 7.757$ Mpa

By solving problem analytically, we get the value of hoop stress equal to 7.757 Mpa.

Results from ansys (σ1-Hoop stress)

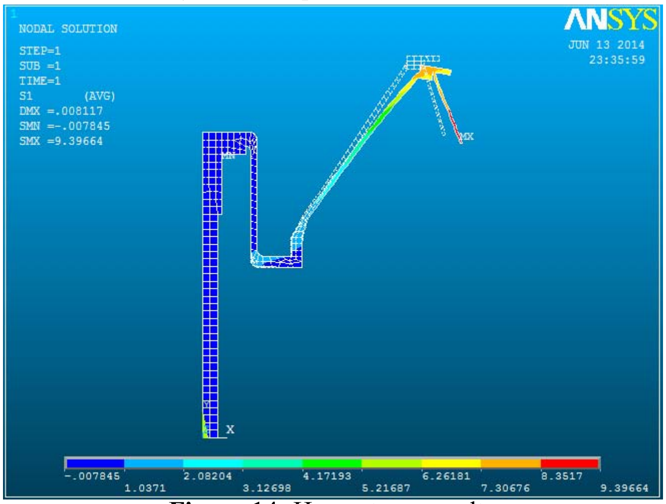

Figure 14: Hoop stress results

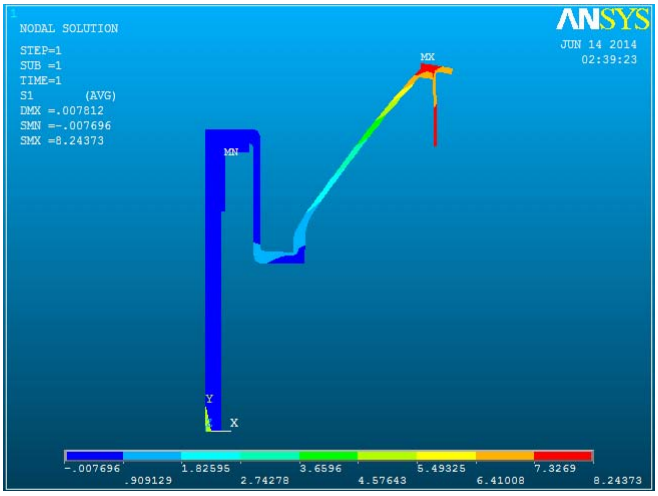

Figure 15: Hoop stress results (Outer bucket straightened)

By solving the same problem from Ansys, we get the value of hoop stress equal to 9.4 Mpa (Figure 14). And to make the calculation more accurate the outer bucket is straightened to get the near values, the hoop stress value for straightened bucket is 8.2 Mpa (Figure 15). The values obtained from finite element method (Ansys) are close to the analytical predictions. The difference in values may be attributed to many factors like analytical predictions, use of different meshes, elements, etc.

Table 4: Analytical and Finite element results

Analytical Result	FEM Result	FEM Result (Outer bucket straightened)
7.757 MPa	9.4 Mpa	8.2 Mpa

8. Conclusion

In this dissertation work, the ANSYS APDL program is written for the analysis of centrifuge structure used in sugarcane plants. The program is developed in such a manner that it can be used for the analysis of all centrifuges of different sizes and speed; more or less the structure will be same. And this method of analysis has the following advantages.

Lead time saving for designers (hand calculations take too much time). We can study limiting speed in a short time. Program helps in improving industrial productivity. By using this program we can study mode shape for different natural frequencies in no time. This method can be used for any other computational gear forces, etc to optimize.

9. Scope of Future Work

The method of finite element analysis by ANSYS APDL can also be extended to other parts of the centrifuge assembly for analysis and comparison to determine the maximum working stresses of them.

References

[1] Boiangiu, M[Ihail] & Boiangiu, A[Dina], “On the vibrations of a centrifuge with the basket in console”, Annals of DAAAM for 2012 & Proceedings of the 23rd International DAAAM Symposium, Volume 23, No.1, ISSN 2304-1382, ISBN 978-3-901509-91-9.
[2] Calin I. Anghel, Radu I Iatan and Gh. Dan Pasat, “Theoretical and experimental studies on disks centrifugal separators”, Periodica Polytechnica Ser. Mech. Eng. Vol. 42, No. 2, 1998, pp 103-116.
[3] Calin Ioan Anghel, “Stress concentration and fatigue life prediction for disks centrifugal separators”, Periodica Polytechnica Ser. Chem. Eng. Vol. 43, No. 2, 1999, pp 117–136.
[4] Mihail BOIANGIU, Georgeta BOIANGIU, “Influence of the constructive and operational features on the natural frequencies of the centrifuges”, RJAV Vol. 8, issue 1/2011, ISSN 1584-7284.
[5] Tadeusz Majewski, Dariusz Szwedowicz, Alejandro Herrera Riquelme, “Automatic elimination of vibrations for a centrifuge”, Elsevier (Science Direct), Mechanism and Machine Theory 46 (2011), pp 344–357.
[6] Liuchuang Wei, Heming Cheng, Jianyun Li, “Modal Analysis of a Cable-stayed Bridge”, Elsevier (Science Direct), Procedia Engineering 31 (2012), pp 481 – 486.
[7] Mohammad Haghpanahi, Reza Mapar, “Development of a Parametric Finite Element Model of Lower Cervical Spine in Sagital Plane”, Proc of the 28th IEEE, 2006, pp 1739-1741.
[8] Xuan Hai-jun, Song Jian, “Failure analysis and optimization design of a centrifuge rotor”, Elsevier

(Science Direct), Engineering Failure Analysis 14 (2007), pp 101–109.

[9] Jiuchen Fan, Yang, Zhaojun, Jixin Wang, Shuwei Ding, “Parametric finite element modeling and nonlinear analysis of vehicle brake”, Proc of IEEE, Aug 2009, ISBN-9781424426935, pp 1762-1766.

[10] B. Gurudat, S. Seetharam, P. S. Sampatkumaran and Vikram Krishna, “Implementation of ansys parametric design language for the determination of critical speeds of a fluid film bearing-supported multi-sectioned rotor with residual unbalance through modal and out-of-balance response analysis”, Proceedings of the World Congress on Engineering 2010, Vol 2 WCE 2010, June 30 - July 2, 2010, London, U.K.

[11] John case, A.H.Chilver and Carl T.F. Ross, “Strength of Materials and Structures”, Arnold-a member of the hodder headline group, London, Fourth Edition, 1999, ISBN-0340719206

Author Profile

Manjunath Kallur received the B.E. and pursuing M.Tech Degrees in Mechanical Engineering from B.L.D.E.A’S DR. P.G.Halakatti College Of Engg & Tech, Bijapur and Mechanical machine design form KLS Gogte Institute of Technology in 2012 and 2014, respectively.

S K Ambekar received the B.E., M.Tech and MBA degrees in Mechanical Engineering from Karnataka University Dharwad, Mechanical Design Engg from IIT Delhi and Master of Business Administration (HR) from Manipal University in 1984, 1991 and 2010, respectively. Currently working as professor in KLS Gogte institute of Technology.