Available online at www.ijiere.com

International Journal of Innovative and Emerging Research in Engineering

e-ISSN: 2394 - 3343 p-ISSN: 2394 - 5494

Integrity Check and Vibration Study for Agitator Vessel by FEA

Harshad Narvekar[a], Vijay Bhosale[a]
[a] K. J. Somaiya College of Engg., Vidyavihar, Mumbai, India

ABSTRACT:
Industrial agitators are machines used in industries that process products in the chemical, food, pharmaceutical and cosmetic industries, in a view of mixing liquids together, promote the reactions of chemical substance, keeping homogeneous liquid bulk during storage or increase heat transfer (heating or cooling). The alarming drawback of the agitator vessel under consideration is its vigorous and unwanted vibration. It is required to check the integrity of the vessel for various operating conditions and also to check the level of vibrations so as to impart modifications if necessary to keep the vibrations of the vessel within the limits from the point of view of proper functioning of the vessel and safety issues. This report consists of structural analysis of equipment using ANSYS software to check the integrity of the vessel and to check the vibration level of agitator shaft.

Keywords: agitator vessel, integrity, structural analysis, vibration

I. Introduction

The agitation in mechanical agitator vessels (pressure vessel) is achieved by the rotation of an impeller which can help in blending, enhancement of heat transfer or enhancement of mass transfer of fluids. The stirred tank reactor with rotating mixers remains the backbone of the chemical process industries (CPI). It is important for the process developers and plant design engineers to understand the mechanical design aspects of agitator reactors. For a successful agitator equipment design, it is necessary to find the ideal balance between the process requirements and an economical, mechanical solution of the vessel-agitator system. [1]

The objective of this project is to find out the root cause of vibration and the feasible solution to minimize it in conjunction with integrity of the vessel using FEA.

The aim of the project is to prepare a CAD model of the equipment as per the specifications using SolidWorks 2012. Static Structural Analysis is performed on the model using ANSYS APDL 14.0 platform to find the von-mises stress and displacement vector sum to check whether the results are within the limits as per ASME Boiler and Pressure Vessel codes VIII Division 1 and 2, Addenda 2011a. Modal Analysis is performed on shaft-impeller assembly to check the vibration levels.

II. CAD Modelling

A. Specifications of the agitator vessel

Table 1: Specifications of the agitator vessel

Sr. No	Description	Qty.	Material	Size (mm)
1.	Main Shell	1	PLATE-SA240-316L SS316L	10712 X 3000 X 10
2.	Upperdish end	1	SS316L	Ellipsoidal 2:1
3.	Bottomdish end	1	SS316L	Ellipsoidal 2:1
4.	Nozzle pad	1	FL-SA182M-F316L	OD: 594.9 ID: 406.
5.	Jacket Shell	1	PLATE SA516GR70	11052 X 2912 X 6
6.	Jacket Bottomdish end	1	SA516GR70	Ellipsoidal 2:1
7.	Pad Plates	4	PLATE SA516GR70	500 X 10
8.	Leg Support	4	PIPE-SA106M-GRB-200DN	OD: 219.08 ID: 174.64
9.	Base plate for leg support	4	PLATE SA516GR70	420 X 420 X 55
10.	Covering for Jacket shell	1	PLATE SA240-316L	OD: 1762 ID: 1710

B. CAD Model

Using the detail drawing and assembly drawing of the various significant parts of the equipment necessary for the analysis are modelled in SolidWorks 2012. The parasolid files of the parts are imported in ANSYS APDL 14.0 platform for further analysis.

Fig. 1 CAD Model of the equipment

III. STRUCTURAL ANALYSIS IN ANSYS APDL

A. Pre-Processing

The assembly of the model is applied with the necessary pre-processor attributes. [2]

1. Type of analysis: Structural & Thermal Analysis
2. Type of element: SOLID186, MASS21, TARGE170, CONTA175
3. The properties imparted to the material model are
   a. Modulus of Elasticity = 2.1e11 N/m^2 at 298 (K)
      Modulus of Elasticity = 1.98e11 N/m^2 at 373(K)
   b. Poisson's ratio = 0.3
   c. Density = 7750 (kg/m^3)
   d. Co-eff. of Thermal Expansion= 1.65E-05 (/K)
   e. Meshing details
      Method: Hexahedral
      Type: Sweep Mesh
      Attributes: Volumes

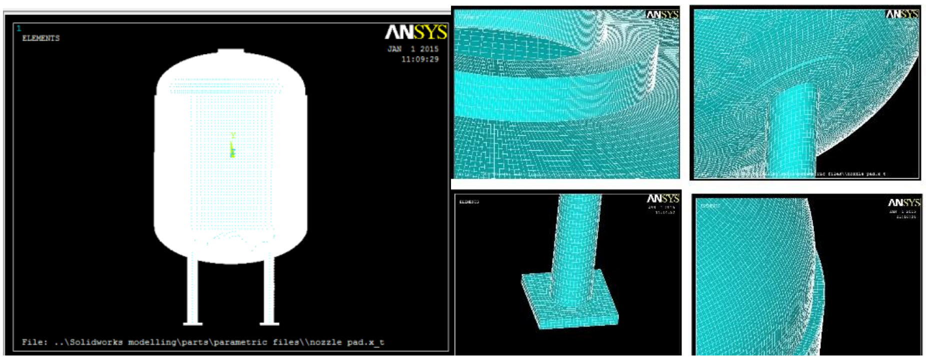


Fig. 2 Matching of nodes in meshed components

B. Solution (Processing in ANSYS)

The meshed model is subjected to various boundary condtions and operating conditions.

Boundary conditions:

1. Fixed support at the base
2. Internal and External pressure wherever applicable
3. Temperatures on surfaces wherever applicable
4. Heat transfer coeffecient on outermost surface= 10 (Watts/m^{2}K)
5. Weight of the agitator assembly acting on nozzle pad= 3100 (N)
6. Torque acting on nozzle pad= 768 (N-m)
7. Moment acting on nozzle pad= 5665 (N-m)

Operating Conditions:

1. Process fluid: Water
2. Design Pressure Shell Side= 0.15 (barg)
3. Design Pressure Jacket Side= 0.15(barg)
4. Design Temperature Shell Side= 373 (K)
5. Design Temperature Jacket Side= 316 (K)
6. Surrounding Temperature= 298 (K)

The model is solved using APDL Solver to obtain the Solution.

C. *Post-Processing*

The model subjected to above boundary conditions is solved to obtain the solution and required results in the Post-Processor step of the procedure:

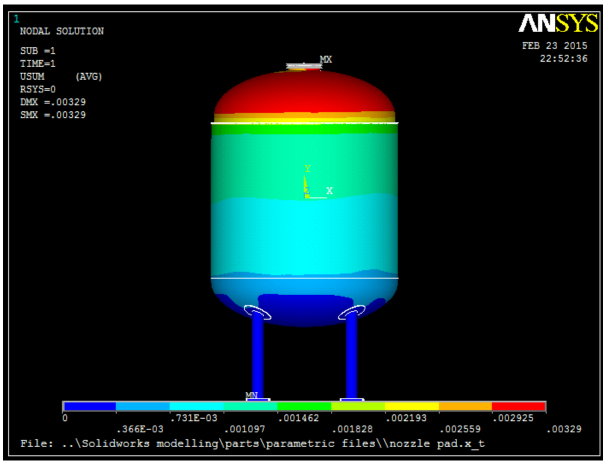


Figure 3: Nodal Displacement Contour plot

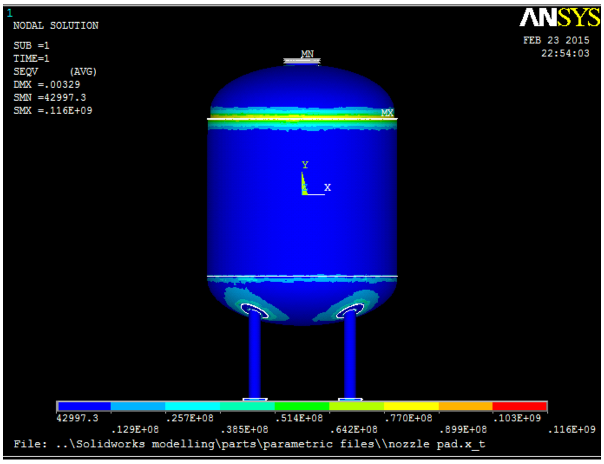


Figure 4: Von Mises Stress Contour plot

i. Von-Mises Stress Calculation

Von-Mises equivalent stress is calculated in each component to check whether the stress generated are within the limits of maximum allowable stress for corresponding materials. The maximum allowable stress values are as specified for materials in ASME BPVC Codes. [3]

Table 2: Von-Mises Stress computed and validation

Sr. No.	Component	Maximum Von Mises Stress from ANSYS (MPa)	Maximum Allowable Stress (ASME) (MPa)	Outcome
1	Main Shell	97.16	115	OK
2	Upper dish end	59.77	115	OK
3	Bottom dish end	96.29	115	OK
4	Nozzle pad	20.08	115	OK
5	Jacket Shell	99.93	138	OK
6	Jacket bottom dish end	60.28	138	OK
7	Jacket Covering	97.35	138	OK
8	Pad plates	20.08	138	OK
9	Leg support	28.39	118	OK
10	Base plates	49.85	138	OK

ii. Dilation of Pressure(agitator) Vessels

The vessel is subjected to pressure, thermal and nozzle loads due to which the components are subjected to strain. Depending upon the nature of stress there occur radial deformations which would increase or decrease the thickness of the vessel. The thickness of the deformed shape should be within the prescribed limits so as to maintain the ovality of the vessel (Table 3) [4]- [5]

Table 3: Nodal displacement values in various components and validation

Sr. No.	Part	Thickness (mm)	Change in thickness (mm)	Allowed change as per ASME (mm)	Outcome
1	Shell	10	0.026	0.1	OK
2	Top dish	6.766	0.009	0.068	OK
3	Bottom Dish	6.786	0.006	0.068	OK
4	Jacket shell	6	0.02	0.06	OK
5	Jacket bottom	6	0.022	0.06	OK

iii. Check of vessel for Design Pressure

The agitator vessel is checked for bursting pressure by calculating the hoop stress and circumferential stress for the design pressure of the shell. [4]

Table 4: Nomenclature of Stress calculation

Symbol	Description	Units
P	Design Pressure	Pa
R	Mean Radius of Shell	m
t	Thickness of component	m
σ_x	Hoop/Longitudinal Stress	Pa
σ_y	Circumferential Stress	Pa
h	Semi-minor axis of ellipse	m

Hoop stress for shell $\sigma_x = \frac{PR}{2t} = \frac{15000 * 1.705}{2*0.01}$ = 1.278e06 (Pa)

Circumferential stress for shell $\sigma_y = \frac{PR}{2t} = \frac{15000 * 1.705}{0.01}$ = 2.556e06 (Pa)

Hoop and Circumferential stress in ellipsoidal head at the center

$\sigma_x = \sigma_y = \frac{PR^2}{2th} = \frac{15000 * 1.705^2}{2*0.0068*0.854}$ = 2.54e06 (Pa)

Hoop stress at tangent line in ellipsoidal head

$\sigma_x = \frac{PR}{2t} = \frac{15000 * 1.705}{2*0.0068}$ = 1.88e06 (Pa)

Circumferential stress at tangent line of ellipsoidal head

$\sigma_y = \frac{PR}{t}\left(1 - \frac{R^2}{2h^2}\right) = \frac{15000 * 1.7}{0.0068}\left(1 - \frac{1.7^2}{2 * 0.068^2}\right)$ = -3.68e06 (Pa)

The stresses induced in the shell and head components are much below the maximum allowable stress (115e06 Pa) as per ASME Code

iv. Linearization of Stress

The general membrane stress (P_m) and local membrane plus bending stress ($P_L + P_b$) is calculated at various critical thickness/junctions throughout the vessel. This is obtained by calculating the linearized stress across Stress Classification Line (SCL). [6]

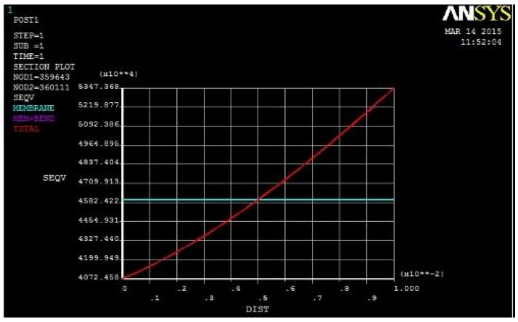


Fig57: SCL-1 across shell thickness

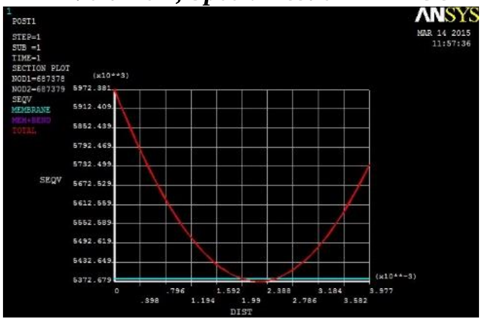


Fig 6: SCL-2 junction of nozzle pad and upperdish

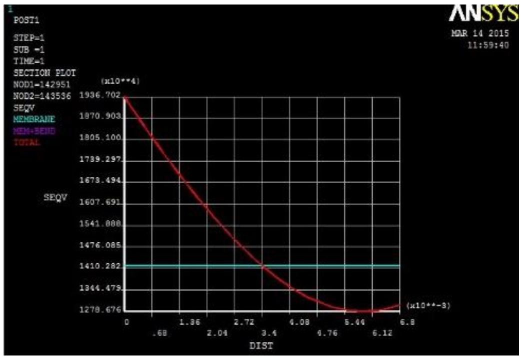


Fig 7: SCL-3 Junction of bottomdish and Shell

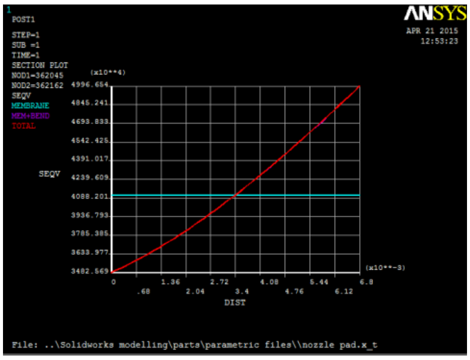


Fig 8: SCL-4 Junction of shell and upperdish

The linearized stress results obtained from SCL are compared with the corresponding allowable stress(S) to evaluate the protection of the vessel against plastic collapse.

Conditions:

$$P_m < S$$
$$P_L < 1.5S$$
$$P_L + P_b < 3S$$

Table 5: Linearized Stress distribution across various sections (Fig 7-10)

	Pm (MPa)	$P_L + P_b$ (MPa)	Allowable (S) (MPa)	Outcome
SCL-1	46.01	53.47	115	OK
SCL-2	5.38	5.972	115	OK
SCL-3	14.19	19.37	115	OK
SCL-4	42.4	49.97	138	OK

v. Theoretical FEA Calculations

The stress and displacement values for few sections are calculated using FEM (Displacement method). Since this method is much time consuming, various computer program are prepared to generate the results as needed. [7]

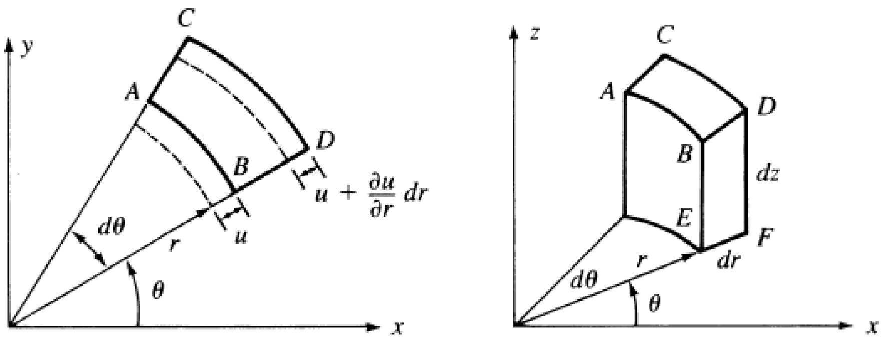


Figure 9: General Element Structure on cylindrical shells [7]

Shear Stress and Strains are zero in r-θ plane. Hence the problem can be considered as a plane-stress/strain problem.

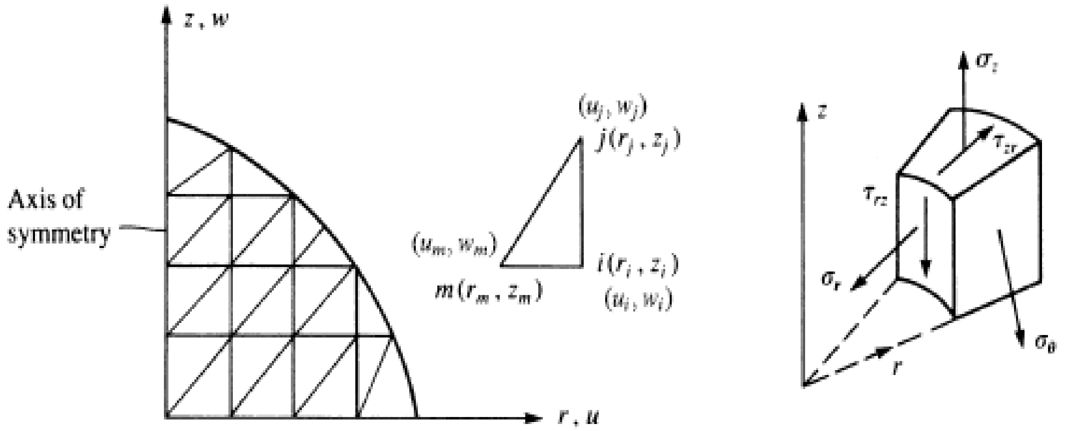


Figure 10: Nodal notations and stress in axisymmetric element [7]

Here we shall consider triangular element consisting of 3 nodes (i, j, m) with two degree of freedom per node.

The nodal displacement is given as:

$$[d] = \begin{bmatrix} d_i \\ d_j \\ d_m \end{bmatrix} = \begin{bmatrix} u_o \\ w_i \\ u_j \\ w_j \\ u_m \\ w_m \end{bmatrix}$$

Using linear displacement function we obtain Strain matrix as

$$[B] = \frac{1}{2A}\begin{bmatrix} \beta_i & 0 & \beta_j & 0 & \beta_m & 0 \\ 0 & \gamma_i & 0 & \gamma_j & 0 & \gamma_m \\ \frac{\alpha_i}{r} + \beta + \frac{\gamma_{i*Z}}{r} & 0 & \frac{\alpha_j}{r} + \beta + \frac{\gamma_{j*z}}{r} & 0 & \frac{\alpha_m}{r} + \beta + \frac{\gamma_{m*Z}}{r} & 0 \\ \gamma_i & \beta_i & \gamma_j & \beta_j & \gamma_m & \beta_m \end{bmatrix}$$

Constitute matrix is given as:

$$[D] = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1-\nu & \nu & \nu & 0 \\ \nu & 1-\nu & \nu & 0 \\ \nu & \nu & 1-\nu & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix}$$

Stiffness matrix is given as:
$K = (2\pi rA)*[B]^T[D][B]$
Assembling the matrix for each element we can obtain a global stiffness matrix.
Using the boundary conditions i.e. surface loads, body loads, etc. we can obtain displacement matrix using:
$[d] = [K]^{-1}[F]$
Using the displacement values we can calculate the Stress values for elements as
$[\sigma] = [D][B]\{d\}$

<u>Stress calculation at Junction of Shell and Upperdish end</u>

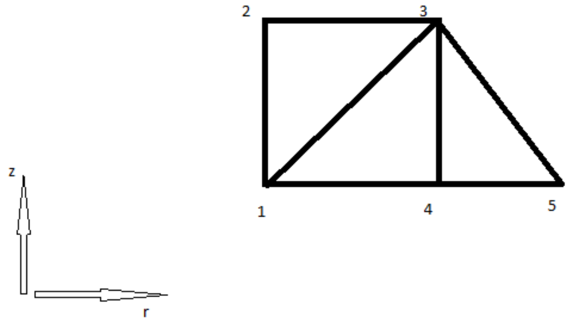


Figure 11: Position of elements considered along the thickness of shell

Table 6: Location of nodes from global axis

Node	r (m)	z (m)
1	1.70	1.4906
2	1.71	1.5
3	1.7068	1.5
4	1.7068	1.4906
5	1.71	1.4906

By using the Finite Displacement method for the above elements we obtain its Global Stiffness matrix as follows:

$$K = (33.17e07) * \begin{bmatrix} 7925 & -2 & -928 & 1281 & -36 & -3451 & -6537 & 2173 & 0 & 0 \\ -2 & 5004 & 1915 & -3237 & -3195 & 6 & 1278 & -1767 & 0 & 0 \\ -928 & 1915 & 7102 & -3193 & -6181 & 1278 & 0 & 0 & 0 & 0 \\ 1281 & -3237 & -3193 & 5004 & 1917 & -1767 & 0 & 0 & 0 & 0 \\ -36 & -3195 & -6181 & 1917 & 7537 & 22 & -1108 & -20 & 0 & 1277 \\ -3451 & 6 & 1278 & -1767 & 22 & 6494 & 42357 & -4727 & 1916 & 0 \\ -6537 & 1278 & 0 & 0 & -1108 & 42357 & 6156269 & -42359 & 111954 & -1277 \\ 2173 & -1767 & 0 & 0 & -20 & -4727 & -42359 & 10325 & -1916 & -3831 \\ 0 & 0 & 0 & 0 & 0 & 1916 & 111954 & -1916 & 13196 & 0 \\ 0 & 0 & 0 & 0 & 1277 & 0 & -1277 & -3831 & 0 & 3831 \end{bmatrix}$$

Applying boundary conditions
F_{1r} = 770N, F_{1z} = 8781.6N
F_{2r} = 770N, F_{2z} = -14830N
F_{3r} = 0 N, F_{3z} = -14723N
F_{4r} = 0, F_{4z} = 8781.6N
F_{5r} = 0N, F_{5z} = 8781.6N

We have obtained stress values in radial, circumferential and longitudinal directions. Using this we can calculate the von-mises stress values in all 3 elements which are obtained as follows:
σ_1 = 5.513e06 Pa
σ_2 = 29.06e06 Pa
σ_3 = 27.1e05 Pa
The average stress value of the element (from $\sigma_1, \sigma_2, \sigma_3$) over the thickness of shell is 20.563e06 Pa.
The average value obtained from ANSYS over the thickness is 40.88e06 Pa (Refer Fig. 10 SCL- 4)
The stress values obtained in both cases are within the maximum allowable stress (1150e05 Pa) for the material as per ASME Values.

vi. Modal Analysis

When the shaft rotates in the agitator vessel, due to the deflections in various points along the length the shaft shall undergo vibration. The speed at which the shaft vibrates violently is known as critical speed. It is highly recommended that range of speed or frequency at which the shaft vibrates should not be within the range of 70% and 130% of the critical speed or natural frequency respectively [8]. Hence the determination of natural frequency and modes of vibration for the undamped shaft-impeller assembly is important which is carried out in ANSYS as follows.

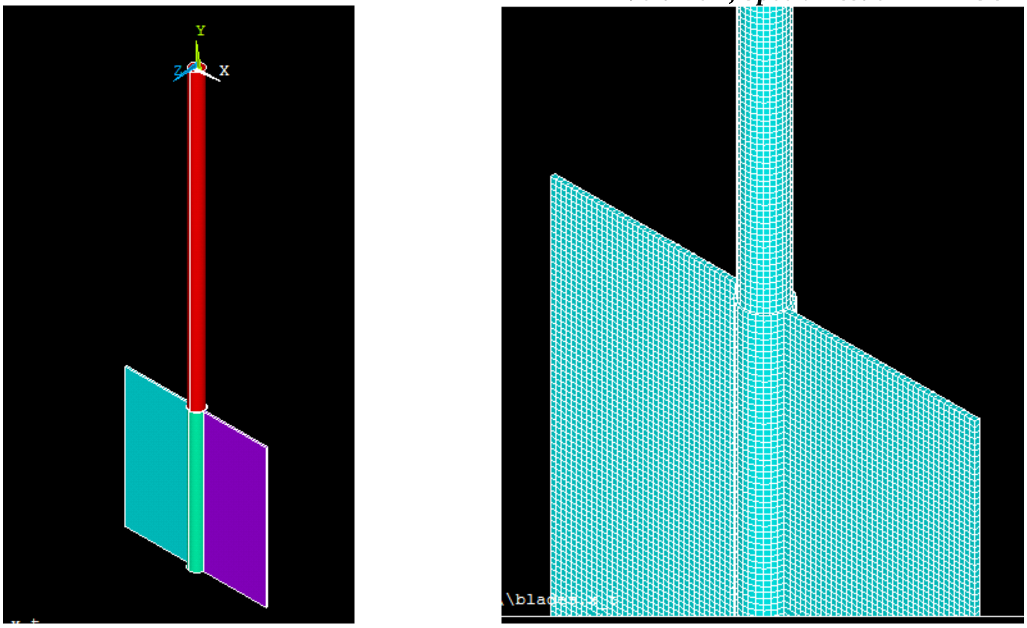
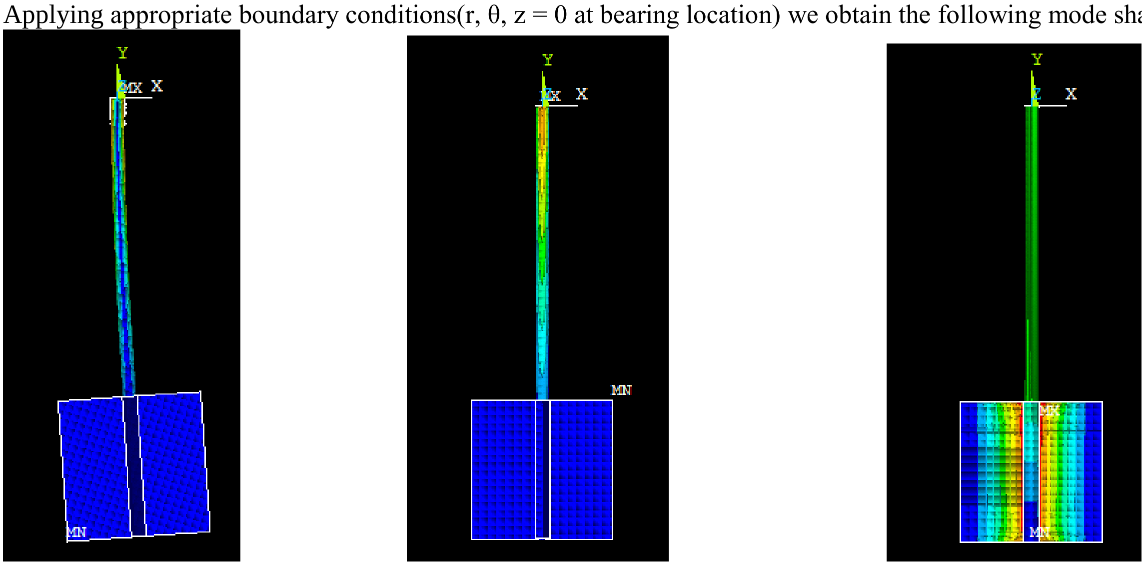


Figure 15: Shaft-Impeller assembly model and Mesh in ANSYS

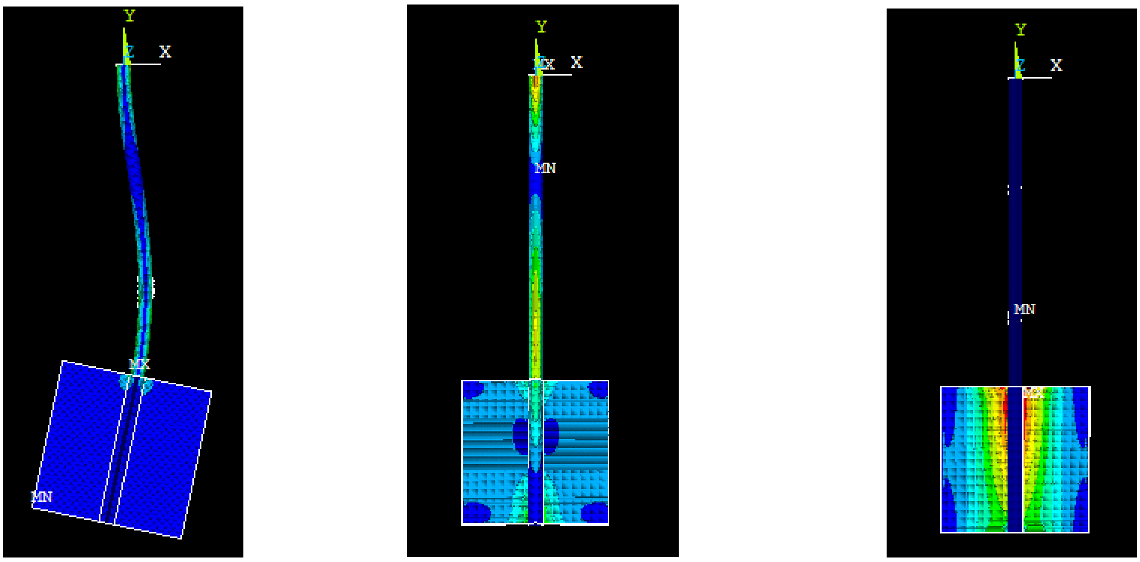
Table 7: Details of Shaft and Impeller

Details	Shaft	Impeller
Material	SA-479 Type 321	SA-240 Type 321
Density(kg/m^3)	8030	8030
Poisson's ratio	0.31	0.31
Dimensions	Diameter = 0.1524m Length = 5m	2 blade gated type 1.6m X 1.6m X 0.018m

Maximum Rotation speed: 42rpm
Element Type: SOLID 186- 20 node Structural solid
Method: Hexahedral Meshing
Type: Sweep meshing
Element attribute: Volumes
Element Size: 0.02m
Number of modes =6
Solving method: Block Lanczos
Applying appropriate boundary conditions(r, θ, z = 0 at bearing location) we obtain the following mode shapes



1st mode 2nd mode 3rd mode



4th mode 5th mode 6th mode

Figure 16: 6 Modes of vibration of Shaft-impeller assembly

Table 8: Natural frequency for 6 mode shapes

Mode number	Frequency(Hz)
1	2.7407
2	2.7514
3	15.1383
4	22.1708
5	22.4054
6	32.8465

The frequency of vibration for rotating elements is given by the cycles it performs in one second.

Hence frequency of vibration of Shaft-impeller assembly = 42/60 = 0.7 cycles/second = 0.7 Hz

We see that the value of operating frequency falls below first natural frequency of vibration.

Range of frequency to avoid resonance = (70% to 130%)*2.7514 = 1.926Hz to 3.577Hz

Hence from the above values it can be seen that the frequency of vibration is not within the range of critical speed. Therefore the upperdish end in contact with the agitator assembly will not vibrate with resonance.

IV. CONCLUSIONS

1. The Stress induced in various components of the agitator vessel are computed using ANSYS and are found to be below the maximum allowable stress values as per ASME.
2. The changes in the dimensions of the vessel confirm to the shape of the vessel.
3. Since the vessel do not fail for the operating conditions, the integrity of the vessel is intact and is safe to operate.
4. Since the shaft does not vibrate with resonance, the vibration of shaft is not responsible for the excessive vibration of the upperdish end.

ACKNOWLEDGMENT

I am highly grateful to Prof. Sangeeta Bansode, K.J. Somaiya College of Engineering and Mr. Riteshkumar Jain & Mr. N. Kulkarni, Toyo Engg. India Ltd, Kanjurmarg for providing the opportunity to carry out my project of Integrity check and Vibration Study of Agitator Vessel by FEA. I would like to express my gratitude to other faculty members of Mechanical Engineering Department of KJSCE, family and friends for providing academic inputs, guidance and encouragement throughout this period.

REFERENCES

[1] M. Stadtaus, H. Weiss, W. Himmelshbach and J. Smith , “Tapping into the mechanics of mixing,” *Chemical Engineering,* vol. 117, no. April 2010, April 2010.

[2] ANSYS INC, “ANSYS Help Topics,” 2011. [Online]. Available: www.ansys.com.

[3] ASME Boiler and Pressure Vessel Committee on Materials, “Properties(Metric) Materials,” 2011.

[4] D. Moss and M. Basic, Pressure Vessel Design Manual 4th Edition, Waltham: Elsevier INC, 2013, pp. 38-73.

[5] ASME Boiler and Pressure Vessels Committee on Pressure Vessels, “2010 ASME Boiler and Pressure Vessel Code Division 1,” 2011.

[6] ASME Boiler and Pressure Vessel Committee on Pressure vessels, “2010 ASME Boiler and Pressure Vessel Code Division 2,” 2011.

[7] D. L. Logan, A First Course in the Finite Element Method, 4th ed., Chris Carson, 2007, p. 836.

[8] M. Joshi, Process Equipment design, The MacMillan Company of India, 1976.