# A REVIEW ON DESIGN, ANALYSIS & WEIGHT OPTIMIZATION OF ROTARY TABLE PALLET

Mali Prashant.[1],Datir Ajay[2], Pravin Pune[3] ,ShirsaleHarichandra[4],Wagh Naresh,Prof.Darade P.P[5]

[1] *Student, Mechanical Department, SND COLLAGE OF* ENGINEERING & RESEARCH CENTER YEOLA, *NASHIK-423401, Maharashtra, India*
[2] *Student, Mechanical Department, SND COLLAGE OF* ENGINEERING & RESEARCH CENTER YEOLA, *NASHIK-423401, Maharashtra, India*
[3] *Student, Mechanical Department, SND COLLAGE OF* ENGINEERING & RESEARCH CENTER YEOLA, *NASHIK-423401, Maharashtra, India*
[4] *Student, Mechanical Department, SND COLLAGE OF* ENGINEERING & RESEARCH CENTER YEOLA, *NASHIK-423401, Maharashtra, India*
[5]*Asst.Professorin Mechanical Department, SND COLLAGE OF* ENGINEERING & RESEARCH CENTER YEOLA, *NASHIK-423401, Maharashtra, India*

## ABSTRACT

*Rotary tables are mostly used for index parts in defined, angular increments. They can be machined, worked in multiple operations. It consists of a circular steel plate, spindles, a drive system, and pins that hold parts in proper place. Rotary tables have fixed or adjustable indexing angles. The table stops for a specified period of time so that an operation can be performed at each station during every revolution. The supporting bearings of rotary tables determine both the load capacity and accuracy. Re circulating ball bearings are cheaper than Angular contact bearings. But angular contact bearings provide better load capacity and axial stiffness[3]. Cross-roller bearings are also used. For a large sized table with high load capacity, the hydrostatic bearings are also used. Selection of rotary tables requires an analysis of specifications and features. It contains maximum indexing increment, rotational speed, torque obtained at the table top, maximum axial load, maximum applicable machining force, maximum radial load, work table diameter. A variety of features are available. Some table surfaces can be raised or lowered at a controlled angle. Others have more than one rotating work surface. Computer numerically controlled (CNC) devices provide greater accuracy and repeatability. Position encoders are often used to relay the position of the table surface.*

**Keywords:***Rotary table, face pallet, FEA*

## PROBLEM DEFINITION

To find the exact shape of the rotary pallet which has optimum stress & deflection by removing the unwanted material to reduce the weight of rotary table.

Analysis of a rotary table pallet will be done for a certain loading condition of an existing model and stresses as well as deflection will be determined for same model by using finite element analysis.Comparison of the results will be done for the modified pallet obtain by trial and error method, to maintain the allowable stresses and deflection.

## OBJECTIVE

1) Design of critical parts like Pallet considering machining force and weight of the work piece into the account.
2) Optimization of weight of the pallet for particular loading conditions and structural rigidity by analytical method.
3) Structural and Dynamic Analysis of pallet will done using finite element package
4) Validation of the theoretical optimization results with finite element results.

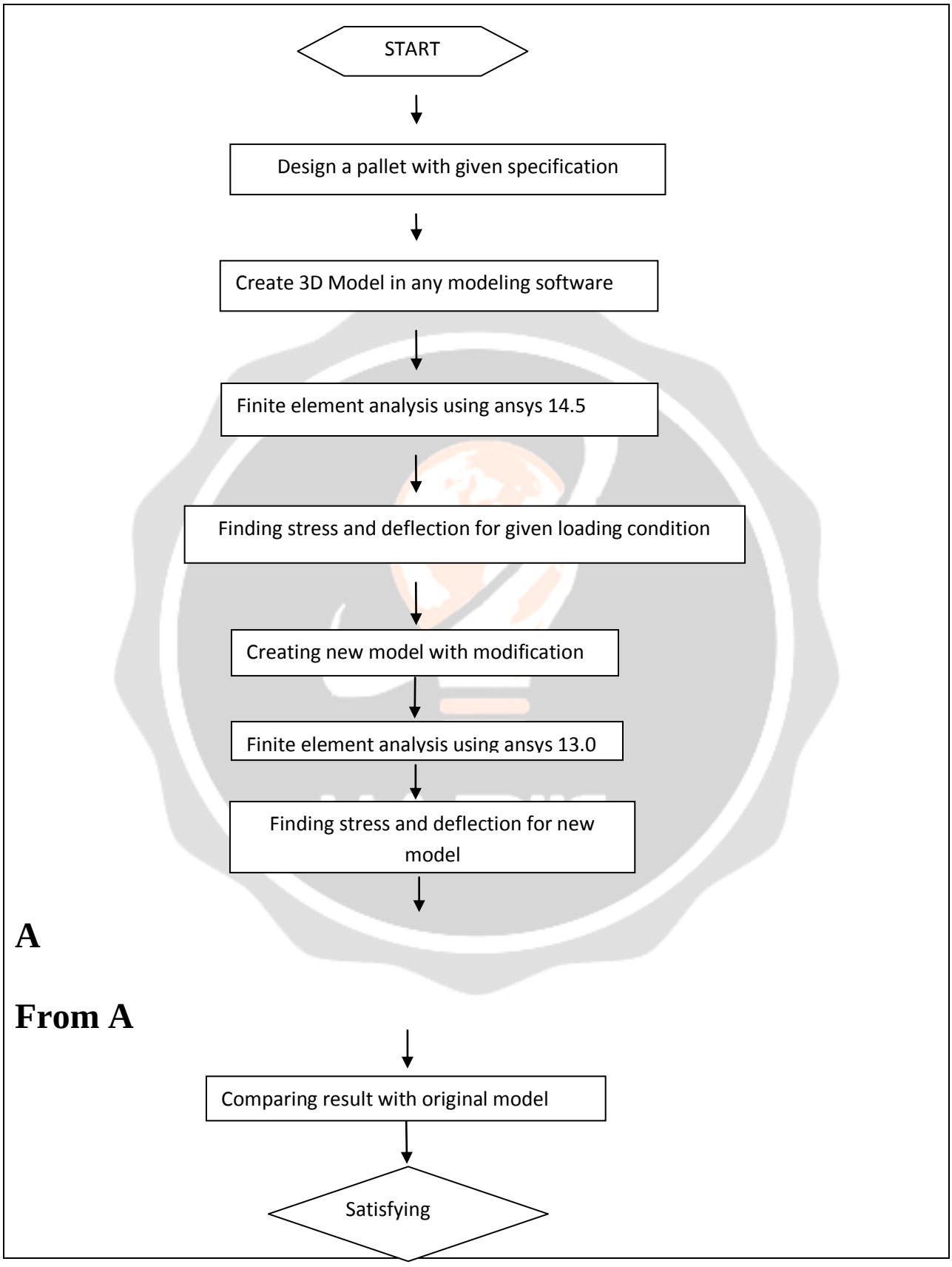
The Dynamic analysis is carried out to check the dynamic behaviour of the critical components like housing and pallet.

## SCOPE

- Removing the unwanted material of the pallet.
- Reduce the weight of rotary table pallet.
- Easy to handle.
- Less machining work require by using steel plates for housing.
- Use of weldment body for housing there is no need of mould and pattern for manufacturing housing.
- Maintain the high machining accuracy.
- Reduce the manufacturing cost.

## METHODOLOGY ADOPTED TO MEET THE OBJECTIVE

The first step is to calculate the parameters by using the given specification. Based on the geometric dimension obtained cad model of the rotary pallet and casing is generated using an appropriate CAD modeling software like SOLIDEDGE, CATIA V5. The generated model is discretized using meshing software Ansys14.5. The discretized models are solved with appropriate boundary condition. The results are obtained are analyzed and required modification to geometric model is carried out and the process is repeated once rotary pallet geometry is optimized.
Initially the pallet is designed for dimensions specified by the customer and it is easily suit the HMC machines for various operations but was not having any considerable deflection. The pallet of size 200 mm diameter is designed for deflection and stresses. The optimized weight of modified pallet is validated by analytical method as well as software. The analysis has been carried out on the existing Pallet by FEA to check the stresses and deflection. The process flow chart to carry out finite element analysis of the rotary pallet.


START
Design a pallet with given specification
Create 3D Model in any modeling software
Finite element analysis using ansys 14.5
Finding stress and deflection for given loading condition
Creating new model with modification
Finite element analysis using ansys 13.0
Finding stress and deflection for new model
A
From A
Comparing result with original model
Satisfying

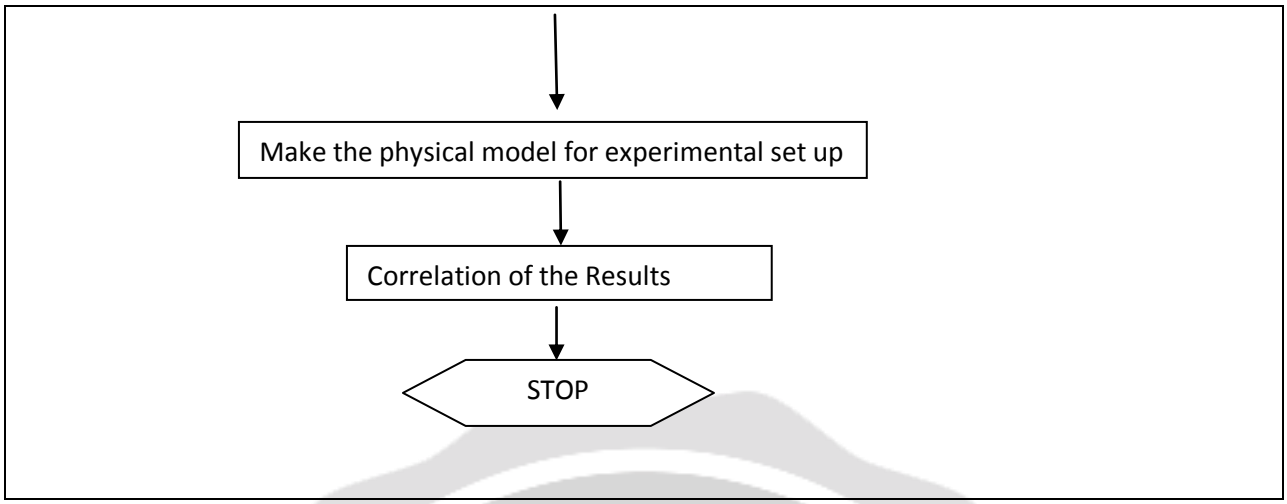



Figure 1 Methodology used in project work

## DESIGN CALCULATION OF EXISTING PALLET

Pallet is the most critical component in the rotary table assembly. Design of the pallet should take care of several aspects like work holding space, less deflection, fixture mounting facilities, T – slots for rigid fastening, specific load carrying capacity, and required elevation, less weight as shown in Figure

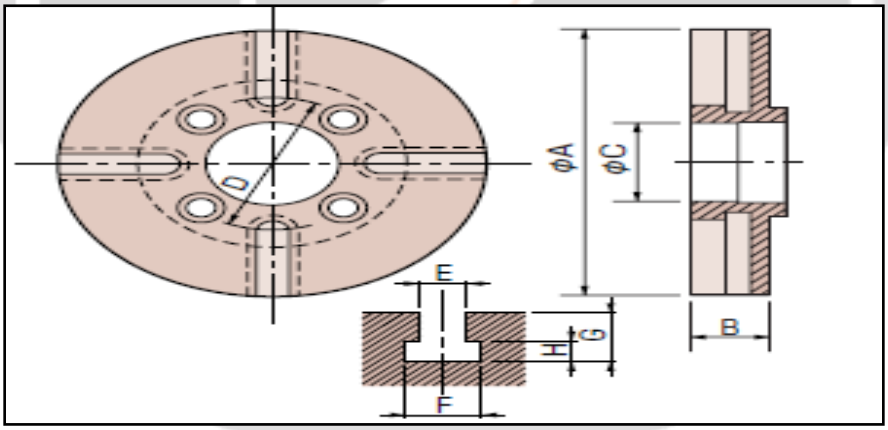



**Figure 2 Details of Rotary Pallet (RNA-200)[1]**

**Table 1 Specification of Rotary table palate from standard catalogue[1]**

Unit : mm

| | A Faceplate outer dia. | B | C | D | E | F | G | H |
|---|---|---|---|---|---|---|---|---|
| RNA-160 | 160<br>200 | 30 | ϕ50H7 | ϕ75 | 12H8 | $19^{+2}_{0}$ | 19 | $8^{+1}_{0}$ |
| RNA-200 | 200<br>250 | 30 | ϕ60H7 | ϕ90 | | | | |
| RNA-250 | 250 | 30 | ϕ75H7 | ϕ110 | | | | |
| RNA-320 | 320 | 40 | ϕ110H7 | ϕ140 | 14H8 | $23^{+2}_{0}$ | 23 | $9^{+1}_{0}$ |

By using the given specifications for RNA 200 shown in Table II, the total force acting on pallet. The material used for pallet is FG300. The Mechanical Properties of FG 300.

Table 2 Mechanical Properties of FG300

| **Sl No** | **Property** | **Unit** | **Value** |
|---|---|---|---|
| 1 | Density[ρ] | $Kg/m^3$ | 7160 |
| 2 | Modulus of Elasticity[E] | GPa | 116 |
| 3 | Poisson's ratio[γ] | NA | 0.29 |
| 4 | Ultimate Tensile Strength[σu] | MPa | 298 |
| 5 | Compressive yield strength | MPa | 972 |
| 6 | Hardness | HRc | 18 |
| 7 | Shear modulus[G] | GPa | 47.6 |
| 8 | Specific Heat | J/kg°c | $0.530 \times 10^{-3}$ |

Total force acting on pallet (F) = Machining force + weight of workpiece + weight of jig &fixture[1]

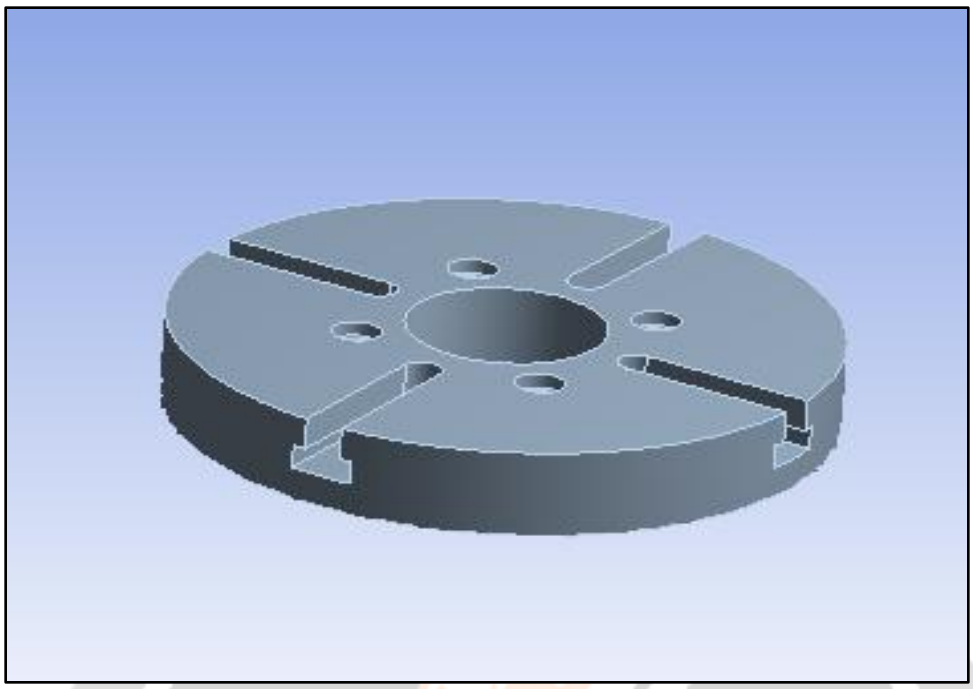


Figure 3. 3D CAD model of Existing pallet (RNA-200)

## BASIC STEPS OF FINITE ELEMENT METHOD

The following sequence of steps describes the solution process that is followed in setting up and solving any equilibrium problem. The following seven steps summarize the finite element analysis procedure.

- **Discretization of the continuum:** Divide the continuum into finite number of sub regions or elements of a simple geometry like 1-D line element, 2-d triangular or quadrilaterals, 3-d tetrahedron etc. such that the original body is modelled as closely as possible. The elements may not be of the same shape and size. At the areas of stress concentration, smaller elements should be used. More number of elements will lead to the exact solution. However this increases the computation cost.
- **Selection of key points:** Selection of key points on the elements to serve as nodes where condition of equilibrium and compatibility are depend is to be enforced. Proper care should be taken at the point of application of load, and the other boundary conditions.
- **Choose proper field variables:** The principal unknown variables of a problem are called field variables. To approximate the distribution of the field variables over the finite element a pattern of field variables in the polynomial form is to be assumed. Finally it is required to find the value of these variables at nodes. Depending upon the problem type, there will be a corresponding field variable model. For example strength of material problems has the displacement model.
- **Generating the system of equations:** For every individual element, the behavioural equations are written. They are usually equation of calculus. These equations may be obtained by various methods like minimum potential energy principle, principle of virtual work etc. for each element a set of equation $\{F\}^e = [K]^e \{u\}^e$ will be used to generate system equation.
- **Globalizing the system equations:** The system equation for each element is usually developed in the form of matrix equations. These individual matrix equations are assembled to obtain a single matrix equation for the total continuum. The component matrices of this final matrix equation are called global matrices.

  A typical final matrix equation will be

$$\{F\}^g = [K]^g \{u\}^g \quad (1)$$

Where

$\{F\}^g$ = global force vector

$[K]^g$ = global stiffness matrix

$\{u\}^g$ = global displacement vector.

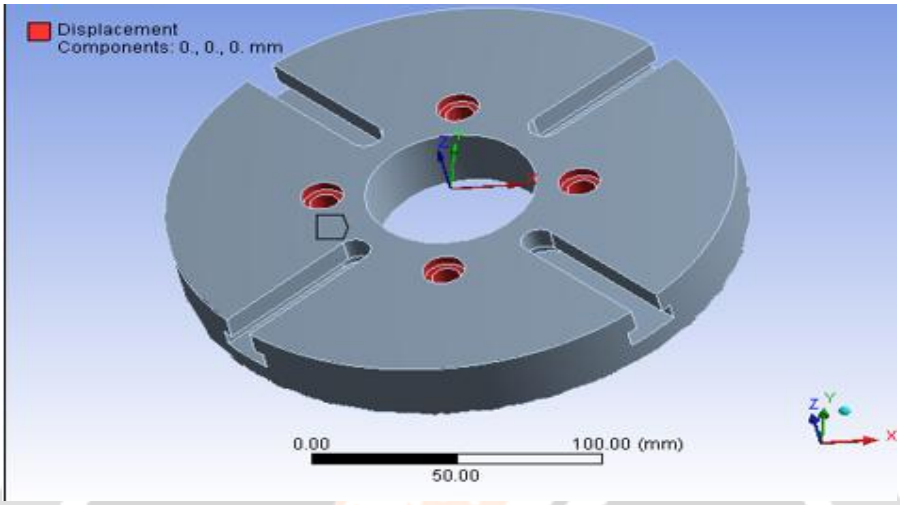



**Figure 4.Applying boundary condition**

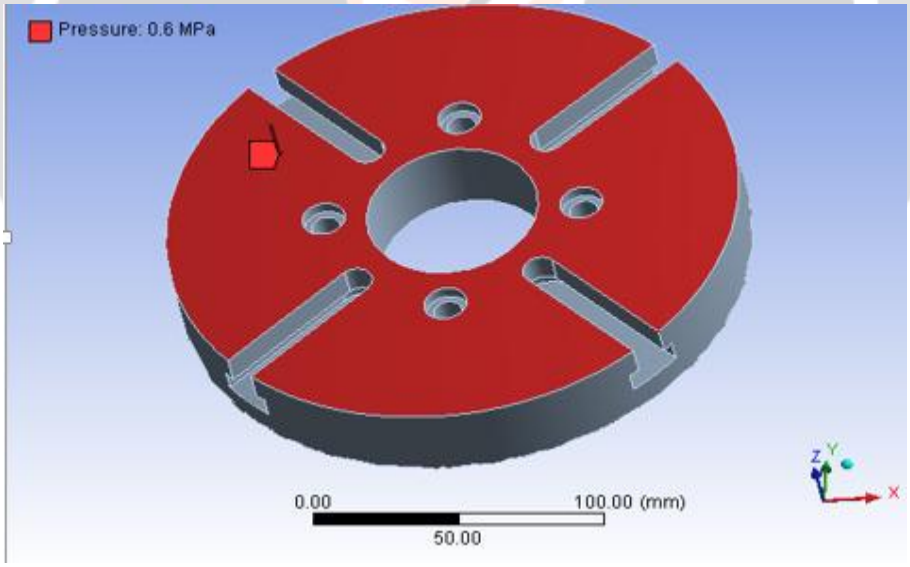



**Figure 5.Applying required meshing**

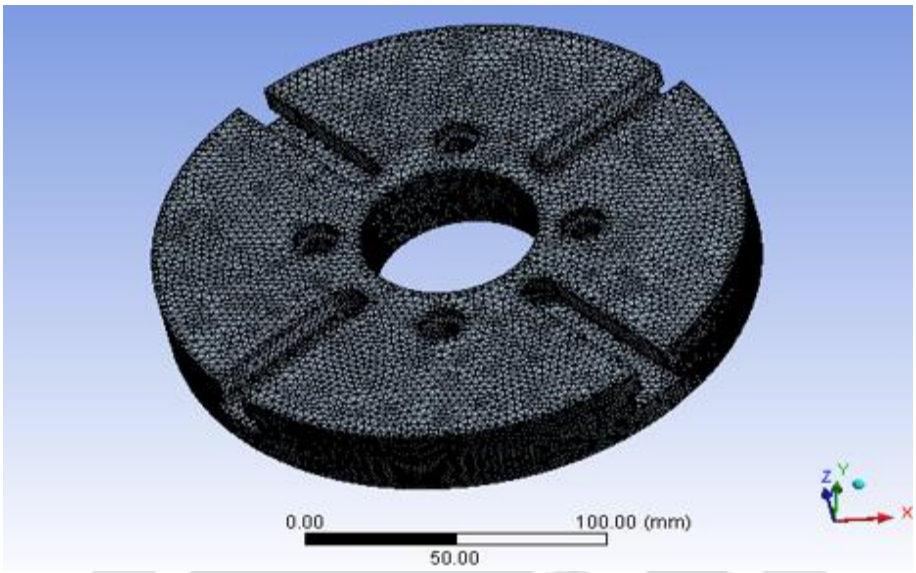



**Figure 6.Meshing of a component**

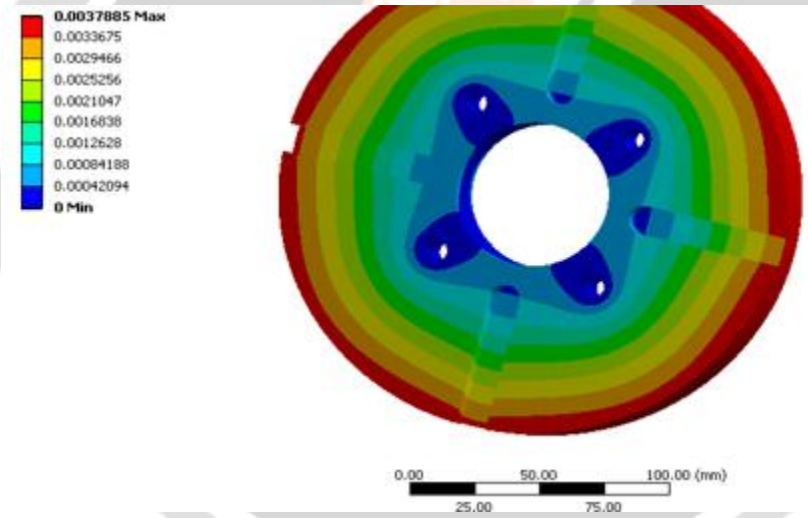



**Figure 7.Result after applying required boundary condition**

After getting the more stress concentration which will hamper design we model new shape from literature survey and starts it FEA analysis.

As the modified pallet design is suitable for fixed clamping support as well as hydraulic clamping support. Now a days, in industries the automatic hydraulic pallet changer of square shape are used, so as per the requirement of industries current set up we design our pallet of square shape with fillet at its all corner.

After modifying the existing pallet design, the total deflection and the stress of the pallet is calculated as follows:

- Length of pallet =200 mm
- Width of pallet =200 mm
- Inner diameter of pallet =60 mm
- Step thickness of pallet =06 mm
- Thickness of pallet =24 mm

- Diameter of small hole =25 mm

Chamfer at its corner =25 mm

The FEA analysis of new component are as below.

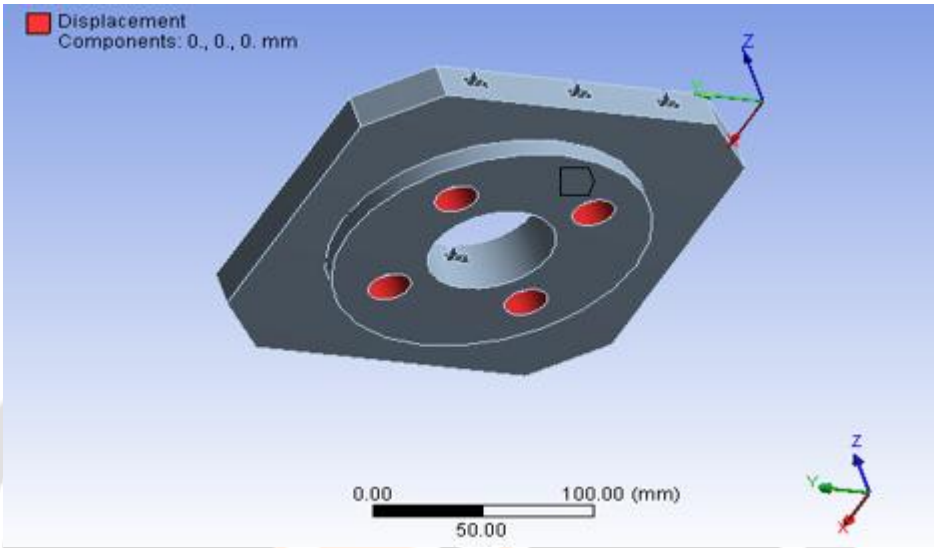



**Figure 8.Applying boundary condition for new model**

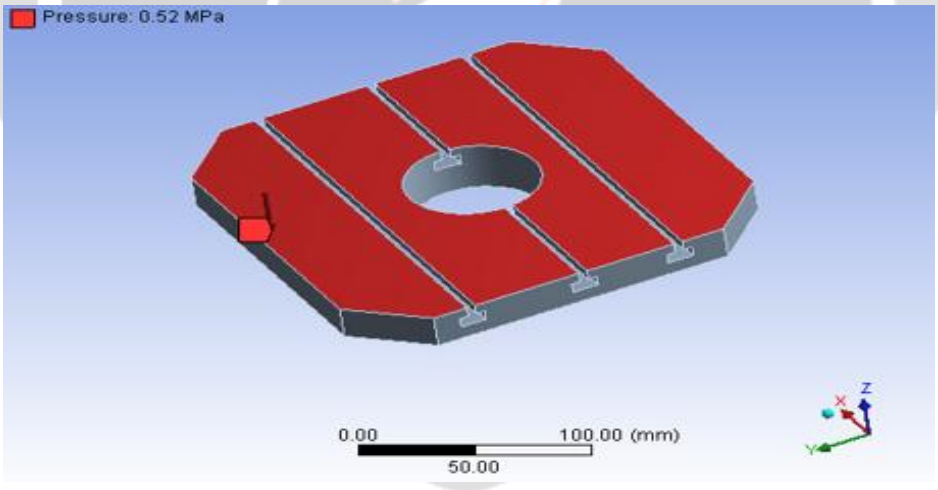



**Figure 9.Applying pressure on new model**

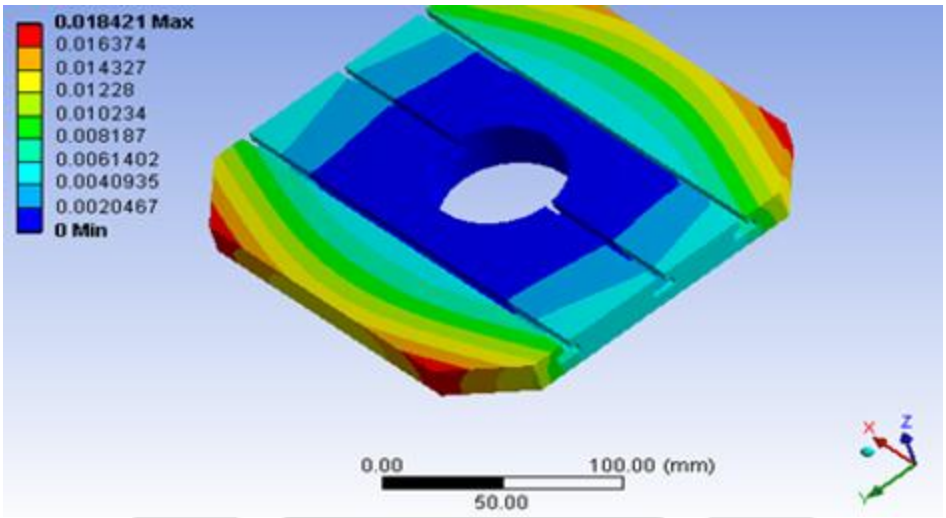



**Figure 10.Structural analysis of new model**

## CONCLUSION

Static structural analysis and modal analysis of the existing and modified pallet is carried out with finite element analysis software ANSYS 15.0 and which gives more safe result for new model.

## REFERENCES:

[1] NianbenZheng, Peng Liu, Zhichun Liu, Wei Liuet, "*Three-fluid cross flow heat exchanger effectiveness revisited"*. J Heat Transf 1995;117:226e9.

[2] Qing Zhang, Huixiong Li , Weiqiang Zhang, Liangxing Li, Xianliang Le, *"Optimization study and heat transfer comparison of staggered circular and elliptic tubes in forced convection*". 2001;44:3953e61.

[3] Wei Wang , Yaning Zhang , Bingxi Li , Huaizhi Han b, XiaoyanGao, *"Finite element analysis for co-current and countercurrent parallel flow three-fluid heat exchanger."*Int J Numer Methods Heat Fluid Flow 2006;16:324e37.

[4] AbdallaGomaa , M.A. Halim and Ashraf Mimi Elsaidet, *"Compound heat transfer enhancement for shell side of double-pipe heat exchanger by helical fins and vortex generators."* Heat Mass Transf 2012;48:1113e24.

[5] www.freefullpdf.com

[6] www.sciecedirect.com

**8.BIOGRAPHY**

| | |
|---|---|
|  | Mr. Mali Prashant, student of the Snd Collage of Engineering & Research Center Yeola, Nashik-423401, Maharashtra, India.in the year 2017-18.In the mechanical Engineering. Work on the project "ROTORY FACE PALLET ANALYSIS USING ANSYS SOFTWARE".<br>Email id-vicky01mahajan@gmail.com |
|  | Mr. Datir Ajay, student of the Snd Collage of Engineering & Research Center Yeola, Nashik-423401, Maharashtra, India.in the year 2017-18.In the mechanical Engineering. Work on the project "ROTORY FACE PALLET ANALYSIS USING ANSYS SOFTWARE".<br>Email id- ajaydatir001@gmail.com |
|  | Mr. ShirsaleHarichandra student of the Snd Collage of Engineering & Research Center Yeola, Nashik-423401, Maharashtra, India.in the year 2017-18.In the mechanical Engineering. Work on the project "ROTORY FACE PALLET ANALYSIS USING ANSYS SOFTWARE".<br>Email id- harichandra.shirsule7@gmail.com |
|  | Mr. WaghNaresh, student of the Snd Collage of Engineering & Research Center Yeola, Nashik-423401, Maharashtra, India.in the year 2017-18.In the mechanical Engineering. Work on the project "ROTORY FACE PALLET ANALYSIS USING ANSYS SOFTWARE".<br>Email id- nareshwagh001@gmail.com |

Prof. DaradeP.P.,.Assistant Professor in mechanical engineering department at SND Collage of Engineering & Research Center Yeola, Nashik-423401,
Email id-pravindarade73@gmail.com