5.1 INTRODUCTION:

The model of turmeric blancher is created on Uni-Graphics™ software for testing and fabrication. The thermal and structural analysis is carried out using ANSYS™ Software. The stainless steel (SS 304L) material is used for fabrication and testing of turmeric mobile blancher. The material used is assumed homogenous and isotropic, there is uniform temperature and heat flow in all direction. Thermal and structural analysis is carried out in the following steps:

- Mesh generation
- Application of boundary and loading conditions
- Review of results

5.2 TYPES OF ELEMENTS AND MESH GENERATION:

Finite element analysis consist of a computer model of design, loaded with different boundary conditions. The design engineer shall be able to verify a proposed design, which is intended to meet design, structure, quality of product, structural failure etc.

Here the given task is divided into number of subdivisions called discretization of the domain and each part is called element. The elements may be line element (1-D), area element (2-D) or volume element (3-D). The elements may be first order linear elements or second order quadratic elements. The volume elements can take the shape of hexahedron (8 nodes), wedge (penta) (6 nodes), tetrahedron (4 nodes) or pyramid (5 nodes), with minimum number of nodes. The number of nodes for higher order (quadratic) elements have an additional number of nodes at midside to get more accurate solution and result. Usually 3D meshing is used when all the three dimensions are comparable. It is applicable for meshing transmission casing, clutch housing, engine block, connecting rod, crank shaft,
hollow shaft etc. SOLID-5, SOLID-45, SHELL-93, SHELL-131, SOLID-226 are the different type of elements used for thermal and structural analysis.

To stimulate conditions properly, correct type of analysis method is to be selected like structural, thermal, fluid, magnetic etc. The model is created in the appropriate units. In the present work the model is created in Uni-Graphics™ software and it is imported in hypermesh™ software for mesh generation. The mesh generation is the process of dividing the analysis into number of discrete finite elements. The mesh may be generated automatically in the hypermesh™ software. Usually automatic mesh generation is preferred for simple geometries and prerequisite is error free CAD model. User has to just select the volume and software automatically carries out meshing as per specified element length, quality criteria etc. The method is very quick with no meshing efforts, but it results in very high number of nodes and elements. The manually created meshing is long and tedious process, it enables the stresses to be captured at the geometric discontinuity and mesh refinement. The precaution to be taken during mesh generation is to work on the fine generation of mesh at region of interest. The regions of high gradients (rapidly changing fields of stress, temperature, pressure etc.) should be identified.

The finite element method has certain requirements on a mesh:

- The mesh must be valid (no holes, self intersections, faces joined at two or more edges).
- The mesh must confirm to the boundary of the domain.
- The density of the mesh must be controllable, the grid density will vary depending on local accuracy requirements.
- In general the shape of the elements should be triangular, rectangular, tetrahedron. The highly distorted elements can lead to numerical stability and causes errors. The quadratic tetrahedral have good finite element method properties and can be generated by using automatic mesh generation algorithms.
- The usual mesh quality parameters to get negative result:
  
a. Warp age is the error occurred usually in hexahedron and shell elements.
b. If the obtuse angle or acute angle between two edges in a quadrilateral is more than the prescribed limit.

c. If the ratio of maximum to minimum length in an element is more than a prescribed limit.

d. If the element is formed with poor shape, size and model point.

Element Description:

In the present work the thermal and structural analysis is carried out by using SOLID-45 for the 3-D modeling of solid structures in ANSYS™. The element is defined by eight nodes having three degrees of freedom at each node with translations in X, Y, and Z directions. The element has plasticity, creep, swelling, stress stiffening, large deflection and large strain capabilities. The geometry, node locations and the coordinate system for this element are as shown in Figure 5.1 with solid stress output.

Figure 5.1 SOLID-45, Geometry and Stress output
The element is defined by eight nodes and the orthographic material properties. The element loads are described in node, pressure may be input as surface load and temperature may be input as element body loads at the nodes. The SOLID-45 element has following advantages:

- Less CPU time is required for element stiffness formation and stress/strain calculations to achieve a comparable accuracy to the full integration option.

- The nonlinear convergence characteristics of the option is generally far superior.

- The analysis will not suffer from volumetric locking caused due to plasticity or any other incompressible material properties.

SHELL - 93 is particularly well suited to model curved shells. The element has six degrees of freedom at each node, has translations in the nodal X, Y and Z directions and rotations about the nodal X, Y and Z axes. The deformation shapes are quadratic and the element has plasticity, stress stiffening, large deflection and large strain capabilities.

The geometry, node locations, coordinate system and stress output for this element are shown in Figure 5.2 and Figure 5.3 respectively. The element is defined by eight nodes, four thicknesses and the orthographical material properties. A triangular shaped element may be formed by defining the same node number for nodes K, L and O. The stress output includes the moments about X face (MX), the moments about Y face (MY) and the twisting moment (MXY). The moments are calculated per unit length in the element coordinate system. The element stress directions and force resultants (NX, MY, TX etc) are parallel to the element coordinate system.

The SHELL-193 element has following advantages:

- The element X and Y axes are in the plane of the element and X-axis may be rotated at an angle toward the Y-axis.
Figure 5.2 SHELL-93, Geometry
The element may have variable thickness, the thickness should not be greater than twice the radius of curvature.

The element has good plasticity, creep, stress stiffening, large deflection, large stain and swelling.

5.3 LOADING AND BOUNDARY CONDITIONS:

After meshing the model, the thermal and structural analysis is carried out by linear static analyzer which is sufficient for situations, where loads are known and the occurrence of peak stress is obvious. Here there is no variation of force with respect to time. The major assumptions of linear static analyzer are:

- All deformation and strains are small.
- Structural deformations are proportional to the load applied.

Figure 5.3 SHELL-93, Stress output
All material behave in a linear elastic fashion.

All loads are static.

No boundary conditions varies with time and application of load.

The linear static analyzer has limitations that the results are violated for non-linear static assumptions, if there is low factor of safety applied to components being designed and if the system behavior is unclear. In linear static analysis the equations are formulated like,

\[ [K] \times [X] = [F] \]

Where,

- \( K \) is stiffness matrix
- \( X \) is the displacement matrix
- \( F \) is the force matrix

The application of correct boundary conditions are critical to get accurate solution of design problem. If this is not taken care, the solution might diverge or we might get junk results.

Based on the process requirements, the turmeric processing machine is designed to handle 50 kg of turmeric in a single batch. The finite element model is generated with the quadrilateral, 2D shell and 3D brick elements to generate the mesh. The self weight is considered 1G (9.81 m/s² and 760 Torr), number of lines are 10082, number of surface areas are 2851, number of 2D elements are 72391, number of 3D elements are 18289 and total number of elements are 90680. Since steam is made to flow through steam receiver, temperature and pressure load is applied on it as shown in Figures 10 and Figure 5.11. The self weight and mass of the turmeric Rhizomes are applied at the centre of the blancher and the steam receiver as shown in Figure 5.12 and Figure 5.13, to avoid development of unbalanced stresses and deflection. The thermal analysis is carried out within temperature limits 120°C to 210°C and pressure limits 3 bar to 9 bar. The analysis is carried out using ANSYS.

5.4 SOLVERS AND REVIEW OF THE RESULTS:

The finite elements solver can be logically divided into three main parts, the pre-solver, the mathematical solver and the post solver. The pre-solver reads in the model created by the preprocessor and formulates the mathematical representation.
of the model. If the model is correct, the solver proceeds to form the element stiffness matrix for the problem and calls the mathematical solver to calculate the primary unknown results like displacement, pressure and temperature etc. The primary results are returned to the solver and the post solver is used to calculate derived results like strains, stresses, heat fluxes, velocities etc. All these results are written to the file which may be read by the post processor. The major sources of errors could be as the problems with units, incorrect loading, improper shaping of linear brick elements, over simplification, discretisation error, in case of large number of finer mesh etc. The formulation error may be due to incorrect assumption, varying displacement field. Numerical errors may be due to problems in numerical accuracy, geometry and loading conditions.

A non linear analysis is usually require to explain the non linearity in stress analysis and to get stress, strain relationship and deformation behavior. Some of the common messages that can interrupt the result and simulation reports:

- The nodes are coincident, but not connected.
- Poisson's ratio is not within the range.
- Shear modulus/young's modulus is not specified.
- Two or more elements share a node, but have incompatible degree of freedom’s.
- An element has a very high aspect ratio.
- Does gravity pull downwards.
- The axial loads influences the stiffness of the structure.
- If there is occurrence of non linear behavior.

5.5 ANALYSIS OF THE PRESENT WORK:

Since the temperature of steam is more than 100\degree C at atmospheric conditions, the blancher thermal analysis is carried out within temperature limits 120\degree C to 210\degree C. Similarly the steam pressure is at least 1.2 bar and it can be increased maximum to 3 bar, it is found that high pressure boilers are not suitable for the plant, hence the pressure limit 1.2 bar to 3 bar is suitable. The thermal analysis is carried out within pressure limits 3 bar to 9 bar using ANSYS\textsuperscript{TM} to operate the blancher safely.
Figure 5.4 and Figure 5.5 show the cylindrical blancher model and Figure 5.6 and Figure 5.7 show the hexagonal blancher model created on Uni-Graphics™ software. Figure 5.8 and Figure 5.9 show the hypermesh model with cylindrical and hexagonal housing respectively. For good aesthetic look the housing may be cylindrical or hexagonal. Figure 5.10, Figure 5.11, Figure 5.12 and Figure 5.13 show the meshed model with load and boundary conditions applied such as temperature, self weight and pressure, mass and all the parameters respectively lumped on the blancher. Figure 5.14 show hypermesh model used for finite element analysis of the blancher with cylindrical housing.

Figure 5.15 and Figure 5.16 show the finite element result for deflection of the blancher with its components. Figure 5.15 show that the deflection of steam pipe is high compared with other components, it may be due to mass of the turmeric lumped into the blancher. For all other parts deflection is minimum. Figure 5.16 show that deflection is high on the cover of the blancher due to improper fitting, but it is within the safe limits.

Figure 5.17 and Figure 5.18 show the finite element analysis result for temperature distribution. Figure 5.17 indicates that the temperature is high in the steam receiver from where steam is distributed in the blancher. Temperatures at all other parts are less and within the limits.

Figure 5.19 and Figure 5.20 show the finite element analysis result for the stresses induced. It indicates that the high stresses are developed on the plummer block and at the hand wheel block since all the load applied is carried by these two blocks supported on stand.

From the results it is found that the temperature of the blancher is within the safe limits, which indicates that there is uniform temperature and heat distribution throughout the assembly. The results of deflection and stress analysis in the pressure range of 3 bar to 9 bar also show that the deflection and stress values are within the limits. The working values of temperature are between 100°C and 120°C and pressure between 2 bar to 3 bar, hence the design may be considered safe.
Figure 5.4 A mobile blancher with cylindrical casing

Figure 5.5: A Mobile Blancher with parts in Cylindrical Casing
Figure 5.6: A Mobile Blancher with Hexagonal Casing

Figure 5.7: A mobile blancher with parts in hexagonal casing
Figure 5.8: A Mobile Blancher with Cylindrical housing

Figure 5.9: Hypermeshed model of Blancher with Hexagonal Casing
Figure 5.10: Meshed Model with Load and Boundary Conditions
[ Temperature 120 °C in Steam Pipes ]

Figure 5.11: Meshed Model with Load and Boundary Conditions
[ Gravity 1G., Pressure applied on steam pipes ]
Figure 5.12: Mass 50 kg lumped on blancher

Figure 5.13: Meshed Model with Load and Boundary Conditions
[All Loads: Pressure, Gravity 1G, Temperature (120), load 50 kg]
Figure 5.14: FEM analysis and Hypermesh model of the blancher with cylindrical casing
Figure 5.15: Finite Element Analysis Results

Case – 1: temperature 120 °C, pressure 3 bar [Assembly Deflection Plot]

Figure 5.16: Finite Element Analysis Results

Case – 1: temperature, 120 °C pressure 3 bar
[Assembly/Housing Deflection Plot]
Figure 5.17: Finite Element Analysis Results
Case – I: Temperature 120°C, pressure 3 bar
[Assembly/Housing Temperature Distribution Plot]

Figure 5.18: Finite Element Analysis Results
Case – I: Temperature 120°C, pressure 3 bar
[Housing Temperature Distribution Plot]
V – Thermal and structural Analysis

Figure 5.19: Finite Element Analysis Results

Case 1: (Temperature 120 °C, pressure 3 bar)

[Assembly Stress Distribution Plot]

Figure 5.20: Finite Element Analysis Results

Case 1: (Temperature 120 °C, pressure 3 bar)

[Assembly/Housing Stress Distribution Plot]