CHAPTER 6

EXPERIMENTAL AND FINITE ELEMENT SIMULATION STUDIES OF SUPERPLASTIC BOX FORMING

6.1 INTRODUCTION

Superplastic properties are exhibited only under a narrow range of strain rates. Hence, it will be useful to determine the optimum strain rate and the thickness distribution, to minimize the cost and the time involved in the experimental analysis. This is done using the finite element simulation. One of the advantages of superplastic forming is its ability to form complex shapes having sharp edges and corners. The rectangular pan shape is one of the complex shapes chosen for this study, to analyse the thinning effect occurring near the corners of the pan. In the present study, Al 7075 alloy was used as the model material. A suitable die having a rectangular box cavity was fabricated from stainless steel. The treated blank specimen was rigidly clamped in between the upper and lower dies. The temperature was maintained constant at 530°C. The other process parameters, like pressure and thickness were selected, using the Design of experiments technique. The die assembly was placed inside the furnace. Compressed air for forming was supplied at the bottom chamber of the die, for a specified period of time. The formed component was taken out of the die and the thickness distribution along the length wise direction was measured (Kalaichelvan 2005).
The same formation process was attempted in the finite element simulation using ABAQUS software. In the simulation process, visco plastic and creep dependent material properties were applied in two steps. The loading and process parameters obtained from the experiment, were given as input for the finite element analysis. The thickness distribution obtained by this technique was compared with that of the experimental results and simulation results.

6.2 EXPERIMENTAL SETUP

6.2.1 Rectangular Shape Die Assembly and Accessories

The forming die consists of the top and bottom parts. The sheet holding provision is in the bottom part, to hold the thermo-mechanically processed 2mm thick sheet blanks, and the top die is of a rectangular shape. Figure 6.1 shows the experimental setup. Figure 6.2 shows the photograph of the die set. In the experiment, the test piece was clamped between the two parts of the die. The accuracy of the electric furnace was maintained within ± 2°C, by using a controller. Figures 6.3 and 6.4 show the bottom and Top pressure forming dies respectively.

Figure 6.1. Experimental setup
Figure 6.2 photograph of the die set

Figure 6.3 Bottom pressure forming die of the Rectangular pan

Figure 6.4 Top pressure forming die of the Rectangular pan
6.2.2 Design of Experiments

The design of experiments technique was used to determine the optimum process parameters for the superplastic forming of the rectangular pan shape. The factors used in this technique are

i. Pressure

ii. Sheet thickness

iii. Forming time

The levels are the values of the selected factors. The list of the selected factors is given in Table 6.1.

<table>
<thead>
<tr>
<th>Sl.No.</th>
<th>Factors</th>
<th>Level 1</th>
<th>Level 2</th>
<th>Level 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Pressure</td>
<td>0.3 MPa</td>
<td>0.35 MPa</td>
<td>0.4 MPa</td>
</tr>
<tr>
<td>2</td>
<td>Sheet Thickness</td>
<td>1 mm</td>
<td>1.5 mm</td>
<td>2 mm</td>
</tr>
<tr>
<td>3</td>
<td>Forming Time</td>
<td>60 min</td>
<td>90 min</td>
<td>120 min</td>
</tr>
</tbody>
</table>

The design of experiments is the simultaneous evaluation of two or more factors (parameters), for their ability to affect the resultant average or variability of a particular process characteristic. The Design Of Experiments (DOE) process is divided into three phases, which encompass all the experimental approaches. The three phases are given below.

i. The planning phase.

ii. The conducting phase.

iii. The analysis phase.
6.2.2.1 The Planning Phase

The planning phase is the most important phase for the experiment to provide the expected information. The following are the steps involved in this phase.

i. Problem Statement

A statement is developed that clearly and concisely describes the problem. By asking questions like “how” and “when” the problem occurs, the problem is identified and defined.

ii. Stating the Objective of the Experiment

The statement of the experimental objective provides the exit criteria for the experiment. The termination of the experiment is addressed in this step.

iii. Select the Quality Characteristics and Measurement System

This step includes the information regarding the determination of the quality characteristic to measure, and the determination of the appropriate measurement system.

iv. Identify the Factors that Influence the Selected Quality Characteristics

The list of factors to be evaluated in the experiment for their effect on the selected quality characteristics should be determined. The factors are the controlling elements in the design of experiments.
v. Select the Levels for the Factors

The values of all the levels of the selected factors and the number of levels for each factor should be determined. A minimum of two levels are required to evaluate a factor’s effect on a given quality characteristic.

vi. Select the Appropriate Orthogonal Array

The selection of which orthogonal to use depends predominantly on these items in the order of priority to

- Assign factors to the orthogonal array and locate interaction.
- The number of factors and interactions of interest.
- The number of levels for the factors of interest.
- The desired experimental resolution.

The first two items determine the smallest orthogonal array that is possible to use, but this will automatically be the lowest resolution, and the lowest costing experiment. The commonly used orthogonal arrays are L9, L18 and L27.

6.2.2.2 Conducting phase

The main items to be considered in the logistics of testing are

- Systems to describe the various test combinations in operational terms.
- A plan to identify the results of the experimental trials.
The experiments as tabulated in the array matrix Table 6.2 are conducted in a sequence.

### 6.2.2.3 The analysis phase

The final phase of the design of experiments is to analyze and interpret the experimental results, to determine the objective of the experimentation, viz., determining the optimum process parameters. When all the tests are conducted, decisions must be made concerning which parameters affect the performance of a product or the process. These decisions are made with the assistance of the following analytical techniques:

- Observation method
- Ranking method
- Column effects method
- Plotting method
- Analysis of variance

Some of these methods for determining the influential factors are subjective in nature, and the others are objective decision making tools. The analysis of variance (ANOVA) will be the predominant statistical method used to interpret the experimental data and make the necessary decisions; since this method is most objective. The other methods are considered as supporting and reinforcing techniques.
Orthogonal arrays are selected depending on the number of selected factors and their levels. In this experiment, a standard L9 array is selected, as shown in Table 6.2. After substituting the process parameters, which are also called as factors and levels, the L9 array is shown in Table 6.3.

<table>
<thead>
<tr>
<th>Trial No.</th>
<th>Pressure, (MPa)</th>
<th>Sheet Thickness, (mm)</th>
<th>Forming Time, (min)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>5</td>
<td>2</td>
<td>2</td>
<td>3</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>3</td>
<td>1</td>
</tr>
<tr>
<td>7</td>
<td>3</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>8</td>
<td>3</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>9</td>
<td>3</td>
<td>3</td>
<td>2</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Trial No.</th>
<th>Pressure, (MPa)</th>
<th>Sheet Thickness, (mm)</th>
<th>Forming Time, (minutes)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.3</td>
<td>1</td>
<td>60</td>
</tr>
<tr>
<td>2</td>
<td>0.3</td>
<td>1.5</td>
<td>90</td>
</tr>
<tr>
<td>3</td>
<td>0.3</td>
<td>2</td>
<td>120</td>
</tr>
<tr>
<td>4</td>
<td>0.35</td>
<td>1</td>
<td>90</td>
</tr>
<tr>
<td>5</td>
<td>0.35</td>
<td>1.5</td>
<td>120</td>
</tr>
<tr>
<td>6</td>
<td>0.35</td>
<td>2</td>
<td>60</td>
</tr>
<tr>
<td>7</td>
<td>0.4</td>
<td>1</td>
<td>120</td>
</tr>
<tr>
<td>8</td>
<td>0.4</td>
<td>1.5</td>
<td>60</td>
</tr>
<tr>
<td>9</td>
<td>0.4</td>
<td>2</td>
<td>90</td>
</tr>
</tbody>
</table>

Table 6.2 Standard L9 array

Table 6.3 Standard L9 array with the process parameters
The observation method is the simplest method of interpreting an experiment with an orthogonal array structure. This effort is focused on those trials having results, that are very similar and having technical appeal. When the most consistent and most desirable group of results have been identified, then the levels of the most important factor will be common for that trial. By simply observing the experimental data, the trial with the most desirable result is observed for the conclusion of the experiment.

6.2.3 Superplastic Forming of the Rectangular Pan

The sheet specimens of thickness 1, 1.5 and 2mm were prepared as per the design of experiments. The specimen was tightly clamped in the die, as shown in Figure 6.1. The die assembly was placed inside an induction furnace, and the temperature was maintained at 530° C. Compressed air at a selected pressure was applied to bottom of the die assembly. The superplastic forming process was completed about the selected forming time.

The experimental work consisted of nine trials, namely, trial 1, 2, 3, 4, 5, 6, 7, 8 and 9; the forming process parameters are listed in Table 6.3. The trial numbers 1, 2 and 6 were not fully formed, the trial numbers 7, 8 and 9 were broken along the sides, and the trial numbers 3, 4 and 5 were formed as per the required shape. The formed component is shown in Figure 6.5. Different places of the thickness measurement in the formed component are shown in Figure 6.6.

![Figure 6.5 Formed component](image)
6.2.4 Measurement of Thickness Distribution

The thickness measurements of the formed components are given in Table 6.4, and the Thinning factors of the formed samples are given in Table 6.5.

Table 6.4 Thickness measurement of the formed component

<table>
<thead>
<tr>
<th>Position</th>
<th>Thickness at different position (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Trial 3</td>
</tr>
<tr>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>2</td>
<td>1.66</td>
</tr>
<tr>
<td>3</td>
<td>1.60</td>
</tr>
<tr>
<td>4</td>
<td>1.65</td>
</tr>
<tr>
<td>5</td>
<td>1.70</td>
</tr>
</tbody>
</table>

Table 6.5 Thinning factor of the formed samples

<table>
<thead>
<tr>
<th>Trial</th>
<th>Average Thickness $t_{avg}$ (mm)</th>
<th>Thickness at Critical(critical) Point $t_{critical}$ (mm)</th>
<th>Thining Factor $(t_{critical}/t_{avg})$</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>1.722</td>
<td>1.60</td>
<td>0.929</td>
</tr>
<tr>
<td>4</td>
<td>0.794</td>
<td>0.69</td>
<td>0.869</td>
</tr>
<tr>
<td>5</td>
<td>1.318</td>
<td>1.16</td>
<td>0.880</td>
</tr>
</tbody>
</table>
By considering the design of experiments, the optimum process parameters have been selected, as: initial sheet thickness of 2mm, operating pressure of 0.3 MPa, and forming time of 120 minutes for this work. The thickness distribution of the deformed component was measured.

In this experiment, trial number 3 of the L9 matrix has the maximum thinning factor and desired height of the rectangular pan. Hence, the factors corresponding to trial number 3 as highlighted in Table 6.3, were taken as the optimum process parameters for forming the rectangular pan.

### 6.3 FINITE ELEMENT METHOD

The Finite Element Method (FEM) has developed into a key, indispensable technology in the modelling and simulation of advanced engineering systems in various fields, like housing, transportation, communications, and so on. In building such advanced engineering systems, engineers and designers go through a sophisticated process of modelling, simulation, visualization, analysis, designing, prototyping, testing, and lastly, fabrication. Note that much work is involved before the fabrication of the final product or system. This is to ensure the workability of the finished product, as well as its cost effectiveness.

The FEM was first used to solve problems of stress analysis, and has since been applied to many other problems like thermal analysis, fluid flow analysis, piezoelectric analysis, and many others. Basically, the analyst seeks to determine the distribution of some field variable, like displacement in the stress analysis, temperature or heat flux in the thermal analysis, electrical charge in the electrical analysis, and so on.
The Finite element method is a numerical procedure for analyzing a wide range of problems, that are too complicated to be solved satisfactorily by classical analytical methods. Since the early 1950s to present day, enormous advancements have been made in the application of the finite element method to solve engineering problems. The Finite element method models a structure as an assemblage of small parts (elements). Each element is of a simple geometry, and therefore, is much easier to analyze than the actual structure.

The general steps involved in the finite element method, with respect to the metal forming analysis using commercial software, are listed below.

1. Create the model of the die and sheet assembly. Based on the feasibility and the shape of the formed product, a choice has to be made between 3 – D, 2-D, axisymmetric or symmetric models. This is essential to obtain accurate results at low computational cost. Complex assemblies can be generated using specialized modeling software, and the model can be imported in to the finite element analysis software as an IGES file.

2. Divide the body in to an equivalent system of finite elements with the associated nodes, and choose the most appropriate element type to model most closely the actual physical behaviour. The accuracy of the result is greatly dependent on the size of the elements. The finer the mesh, greater is the accuracy; however, this increases the computational time. Commercial computer programs, called pre-processors, help in generating a mesh.
3. Formulate the properties of each element.

4. Select the material model to be associated with the die and the sheet. This is essential to determine the strain/displacement and stress/strain relationship during the formulation of the problem.

5. Assign the information regarding the boundary and initial conditions. This is necessary for the solver to evaluate the equations. This involves constraining all the degrees of freedom of the nodes associated with the clamped portion of the sheet and applying the symmetric boundary conditions.

6. Apply the loads associated with the complete finite element model, which must also be provided unless there is no load on the model as in many analyze involving natural frequencies extraction. In the case of metal forming, it involves releasing the plunger, over the clamped sheet. This is usually done by assigning a fixed displacement to the plunger or releasing it with an initial velocity.

7. Solve the problem and interpret the results; for most of the applied mechanics problems, the displacements are the true nodal variables that the solver will compute. The software, however, can also compute the stress and strain from the interpolation of these nodal displacements automatically, and this can be done by specifying them in the input file.

The advantages associated with the finite element analysis are, its ability to model irregular shaped bodies, handle general load conditions, handle unlimited number and kinds of boundary conditions, alter the finite
element model relatively easily and cheaply, include dynamic effects and handle non-linear behavior existing with large deformation and nonlinear materials. These advantages make it an ideal tool for the analysis of the superplastic forming process.

The use of numerical analysis techniques in present day industries; has reduced the trial and error procedure followed earlier. The pressure on the modern day industry is for continuous improvement, which leads to new products being developed very frequently.

The current industrial requirement from the numerical analysis of sheet metal forming is broadly classified as:

- Time reduction
- Cost reduction
- Increase in product quality

For these three requirements to be fulfilled, it is expected that

- The simulation tool is able to model various process and operation of sheet metal forming.
- The simulation tool is user friendly
- There is an CAD-FEM-CAD interface
- The analysis is very efficient
- The various outputs required, such as stress, strain, thickness distribution, failures modes, etc. Should be computed easily.
- And there should be various models present, to represent the material behavior accurately.
6.4 FINITE ELEMENT SIMULATION OF BOX FORMING

The finite element analysis of the superplastic forming of a rectangular pan was performed under the following assumptions

i. The Material is isotropic

ii. Grain growth is negligible

iii. Strain hardening is negligible

iv. The relationship between the flow stress and strain rate is defined by the constitutive equation (Equation 2.2) as mentioned in section 2.2.

In this simulation, a circular blank of 80mm diameter and 2mm thickness, was considered as a starting material for superplastic forming.

The behaviour of a phenomenon in a system depends upon the geometry or domain of the system, the property of the material or medium, and the boundary, initial and loading conditions. For an engineering system, the geometry or domain can be very complex. Further, the boundary and initial conditions can also be complicated. It is therefore, in general, very difficult to solve the governing differential equation via analytical means. In practice, most of the problems are solved using numerical methods. Among these, the methods of domain discretization championed by the FEM are the most popular, due to its practicality and versatility.

The procedure of computational modeling, using the FEM, consists broadly of four steps:
• Modelling of the geometry.
• Meshing (discretization).
• Specification of the material property.
• Specification of the boundary, initial and loading conditions.

6.4.1 Geometric Modeling and Meshing

Superplastic forming involves modelling complex shaped structural components. It is essential to accurately define an intricate shaped die surface, and generate a quality finite element mesh over it, into which the flat sheet metal is deformed. Taking advantage of the symmetry only a quarter of the assembly is considered for the analysis. Usually different types of symmetry may exist in an assembly. These include reflective or mirror, skew, axial and cyclic. Symmetry means correspondence in size, shape and position of loads, material properties, and boundary conditions that are opposite sides of a dividing line or plane. The use of symmetry allows us to consider a reduced problem instead of the actual problem, and the computational time required for the analysis is substantially decreased. One important aspect to be considered during a symmetric analysis is to model more than quarter of the die surface. This prevents the elements and nodes on the deformable sheet from sliding off the die.

Meshing is performed to discretize the geometry created into small pieces, called elements or cells. We can expect the solution for an engineering problem to be very complex, and varied in a way that is very unpredictable, using functions across the whole domain of the problem. If the problem domain can be divided (meshed) into small elements or cells using a set of grids or nodes, the solution is within an element, and can be approximated very easily using simple functions such as polynomials. The solutions for all the elements thus form the solution for the whole problem domain.
Mesh generation is a very important task of the pre-process. It can be a very time consuming task to the analyst, and usually an experienced analyst will produce a more credible mesh for a complex problem. The domain has to be meshed properly into elements of specific shapes, such as triangles and quadrilaterals. Information, such as element connectivity, must be created during the meshing for use later in the formation of the FEM equations.

6.4.1.1 The superplastic alloy sheet

Create individual parts by sketching or importing their geometry. A part created using the part module tools is called a native part and has a feature-based representation.

Parts are the building blocks of the ABAQUS software; each body in the finite element model is associated with a part. The part types are deformable parts, discrete rigid parts, analytical rigid parts and Eulerian parts.

The superplastic sheet is meshed using quadrilateral membrane elements, of the type M3D4 present in the ABAQUS element library. M3D4 in a term used for 3-Dimensional, 4 node membrane element. Computer simulations of three dimensional superplastic sheet forming processes can be carried out by the finite element method, with a membrane element or a shell element. A membrane element is regarded as more preferable rather than a shell element, because of the computing efficiency and the easy contact treatment. Membrane elements are sheets in space that carry membrane force, but do not have any bending or transverse shear stiffness; the membrane is in a state of plane stress. The bending effect can be neglected, because the thickness of the superplastic sheet is negligible, as compared to the other sheet dimensions. Figure 6.7 shows the part module of the sheet.
6.4.1.2 The die surface

The female die is modelled as a rigid body and is meshed with rigid R3D3 elements. The rigid surface is defined with the SURFACE option, by grouping together those faces of the 231 R3D3 elements used to model the die, that face the contact direction and rigid element mesh. Figure 6.8 shows the part module of the die. The section and material definitions are created and assigned to regions of parts in the property module. Figure 6.9 shows the property module of the die, and Figure 6.10 shows the property module of the sheet. Created and assemble the part instances are in the model shows in Figure 6.11. The material in the blank is assumed to be elastic-viscoplastic, and the properties roughly represent the 7075 (Al- 5.5 Zn – 1.5 Cu- 2.6 Mg )-based commercial superplastic aluminum alloy at 530°C. It has a Young's modulus of 70 GPa and a Poisson's ratio of 0.34. The flow stress is assumed to depend on the plastic strain rate according to

\[ \sigma = A (\dot{\varepsilon}_{pl})^{1/2} \]  \hspace{1cm} (6.1)

where,

A is 179 MPa and the time is in seconds (Abaqus Manual 6.5).
Figure 6.8 Part module of die

Figure 6.9 Property module of the die
6.4.2 Creating the Components and Defining Contact Properties

Many engineering systems consist of more than one material. The property of the materials can be defined either for a group of elements or each individual element, if needed. For different phenomena to be simulated,
different sets of material properties are required. For example, the Young’s modulus and shear modulus are required for the stress analysis of solids and structures, whereas the thermal conductivity coefficient will be required for a thermal analysis. The keeping of the Input of a material’s properties into a pre-processor is usually straightforward; all the analyst needs to do is key in the data on the material properties, and specify either to which region of the geometry or which elements the data applies. However, obtaining these properties is not always easy. There are commercially available material databases to choose from, but experiments are usually required to accurately determine the property of the materials to be used in the system.

The contact problem during the SPF process is complex. The Contact between the sheet and the die is highly nonlinear, because of its asymmetry, where at a position in space a node is either free or rigidly constrained, depending on an infinitesimal change of position normal to the die surface. ABAQUS defines the contact between two bodies in terms of two surfaces that may interact; these surfaces are called a contact pair. For each node on the slave surface, the ABAQUS attempts to find the closest point on the master surface of the contact pair, where the master surface’s normal passes through the node on the slave surface. During the analysis it is essential, that the rigid surface must always be the master surface, and the deformable bodies must be the slave surface.

To avoid having points “fall off” the rigid surface during the course of the analysis, more than a quarter of the die has been modelled. It is always a good idea to extend the rigid surface far enough, so that the contacting nodes will not slide off the master surface.
ABAQUS generates a unique normal to the rigid surface at each node point, based on the average of the normal’s to the elements sharing each node. There are times, however, when the normal to the surface should be specified directly.

The blank was assumed to be elastic-visoplastic, and the properties have represented the commercial superplastic 7075 Al alloy with Young’s modulus of 70 GPa and a Poisson’s ratio of 0.3. The material properties were given in terms of two steps: in the first step, the elastic property was given along with the isotropic condition, and in the second step, the viscoplastic property was given for the blank. The step module is created and defines the analysis steps, shows in Figure 6.12. Specify the interaction, shows in Figure 6.13.

Figure 6.12 Step module
6.4.3 Boundary Conditions and Loading Options

Superplasticity is exhibited by materials only in a narrow strain rate range, with an optimum value unique to each material. This factor makes it essential to determine the pressure loading history, in order to maintain the maximum strain rate near the optimum value throughout the whole forming process.

The boundary, initial and loading conditions play a decisive role in solving the simulation. Inputting these conditions is usually done easily, using commercial pre-processors, and it is often interfaced with graphics. Users can specify these conditions either to the geometrical identities (points, lines or curves, surfaces, and solids) or to the elements or grids.

The clamped region of the sheet is represented by constraining all degrees of freedom of the nodes along the periphery of the sheet. Symmetric boundary conditions are applied to the nodes on the plane of symmetry. On the plane of symmetry the displacement in the direction perpendicular to the
plane must be zero. The die surface is completely fixed with respect to all degrees of freedom.

The female die is created as a rigid body, and is meshed with rigid R3D3 elements. The rigid surface is defined with the SURFACE option by grouping together those faces of the 231 R3D3 elements used to model the die. The mechanical interaction between the die and the blank is a frictional contact, having a value of 0.125 as the coefficient of friction. A quarter of the blank was modeled, using 704 membrane elements of type M3D4. These are fully integrated bilinear membrane elements.

The superplastic forming of the rectangular pan was simulated for the optimum process parameters of 0.4 MPa, 2 mm sheet thickness and a forming time of 120 minutes. The thickness distribution of the rectangular pan along the lengthwise direction obtained from the experiment and simulation was compared. The superplastic forming of the rectangular pan (a quadrant) obtained by the simulation.

The load is applied in terms of two steps; the initial application of the pressure is assumed to occur so quickly, that it involves a purely elastic response. This is achieved by using the static procedure. During the VISCO (visco property) step the parameter CETOL (tolerance) controls the time increment, and the accuracy of the transient creep solution. The ABAQUS compares the equivalent creep strain rate at the beginning and the end of an increment. A constant load of 0.3 MPa was applied in both the steps for a constant forming time of 120 minutes. The specific loads, boundary conditions, and fields in the load module are shown in Figure 6.14. The mesh module is to create a finite element mesh as shown in Figure 6.15. Submit a job for analysis and monitor its progress. The Job module is shown in Figure 6.16.
Figure 6.14 Load module

Figure 6.15 Mesh module
6.5 RESULTS AND DISCUSSION

The rectangular pan forming model consists of a sheet 32mm long and 22mm wide, with a thickness of 2mm. This sheet is placed over a box die having a depth of 10mm. The sheet is modelled with the help of 704 quadrilateral membrane elements. 231 rigid elements of the type R3D3 are used to model the female die. The female die is extended along the axis of symmetry, to avoid the contacting nodes from sliding off the master surface. Coulomb’s coefficient of friction is defined between the sliding surface of the die assembly. The sheet is clamped along the circumference, and symmetric boundary conditions are applied along the axis of symmetry.

The superplastic forming of the rectangular pan was simulated, for the optimum process parameters of 0.3 MPa, 2 mm sheet thickness and a forming time of 120 minutes. From the experimental results, the lengthwise thickness distribution was considered as important. The stress distribution, the lengthwise thickness distribution, and the strain on the surface of the
rectangular pan were obtained from the simulation. The stress was maximum at the corners of the rectangular pan, and this revealed that the severe stress was encountered in the corners of the rectangular pan. Figure 6.17 shows the resulting stress in step 1- initial stage; Figure 6.18 shows the resulting stress in step 1- final stage; Figure 6.19 shows the resulting stress in step 2- initial stage, Figure 6.20 shows the resulting stress in step 2- intermediate stage, and Figure 6.21 shows the resulting stress in step 2- final stage.

Figure 6.17 Resulting stress in step 1- initial stage

Figure 6.18 Resulting stress in step 1- Final stage
Figure 6.19 Resulting stress in step 2- initial stage

Figure 6.20 Resulting stress in step 2- intermediate stage

Figure 6.21 Resulting stress in step 2- final stage
Thus, the critical region of the rectangular pan was found to be the corners of the rectangular pan, which have undergone the maximum stress and strain. The extreme thinning occurred near the corners of the rectangular pan. The fully deformed view in Figure 6.21 shows that the maximum surface strain occurs at the corners of the rectangular pan. Figure 6.22 shows the resulting strain in step 1- initial stage, Figure 6.23 shows the resulting strain in step 1- final stage, Figure 6.24 shows the resulting strain in step 2- initial stage, Figure 6.25 shows the resulting strain in step 2- intermediate stage and Figure 6.26 shows the resulting strain in step 2-final stage.

Figure 6.22 Resulting strain in step 1- initial stage

Figure 6.23 Resulting strain in step 1-final stage
Figure 6.24 Resulting strain in step 2 - initial stage

Figure 6.25 Resulting strain in step 2 - intermediate stage

Figure 6.26 Resulting strain in step 2 - final stage
Figure 6.27 shows the resulting displacement in step 1- initial stage, Figure 6.28 shows the resulting displacement in step 1- final stage, and Figure 6.29 shows the resulting displacement in step 2- initial stage. Figure 6.30 shows the resulting displacement in step 2- intermediate stage, and Figure 6.31 shows the resulting displacement in step 2- final stage.
Figure 6.29 Resulting displacements in step 2- initial stage

Figure 6.30 Resulting displacements in step 2- intermediate stage

Figure 6.31 Resulting displacements in step 2- final stage
The thickness distribution of the rectangular pan along the lengthwise direction as seen from the experiment and the simulation, was compared and is shown in Figure 6.32. The simulation results were also in good agreement in this work. Hence, for the various sizes, rectangular or other typical component dimensions, this simulation model can be extended well.

![Figure 6.32 Thickness distribution of the Rectangular pan](image)

**Figure 6.32 Thickness distribution of the Rectangular pan**

### 6.6 SUMMARY

The 7075 Al alloy was selected for the superplastic forming of the rectangular pan. From the design of experiments, the third trial of the L9 array was selected. The sheet thickness of 2 mm was obtained by the thermomechanical treatment. The grain size was ensured to be less than 10µm. The forming pressure of 0.3 MPa, the forming time of 120 minutes and forming temperature 530°C were considered. The experiment was conducted to obtain the required shape of the component. The lengthwise thickness distribution was measured. The finite element model was created,
using ABAQUS software. The thickness distribution of the rectangular pan along the lengthwise direction as seen from the experiment and the simulation, was compared. The rectangular pan made by the experiment was in close agreement with the finite element simulation, and the variation lies within 8%.