Designing and Finite Element Analysis of Tube Hydroforming Setup for Joining of Two Tubes

Sweety Chavda¹ Prof. Chandresh Motka²
¹PG Student ²Assistant Professor
¹,²Department of Mechanical Department, Kalol Institute of Technology and Research Centre Kalol, Gujarat, India

Abstract— Tube hydroforming is a forming method which has several advantages. By using pressure in combination with material feeding it is possible to manufacture products with high structural integration and tight dimensional tolerances. The tube hydroforming industry can benefit from Finite Element simulations, and this simulation methodology and practically investigate the result is the topic of this dissertation. The material testing needs to account for the specifics of tube hydroforming. The process parameters in hydroforming are the inner pressure and the axial feeding, where a correct combination of these parameters is crucial for the success of the process or joining of two tubes of similar material by tube hydroforming process.

Key words: Finite Element Analysis (FEA), Tube Hydroforming Process (THP), Formability, Forming Pressure, Forming Limiting Diagram (FLD)

I. INTRODUCTION

Internal high pressure forming methodology used for the manufacturing of hollow metal component is known as hydroforming process. In this process blanks are expanded from inside out by means of liquid working medium in a closed die. There are two version of hydroforming, tubular and sheet.

The automobile structural industry is involved in a process to minimize cost but taking into account the optimization of product concerning weight and strength. The innovative and most advance hydroforming process, idle for producing the seamless and light weight product is TUBE HYDROFORMING PROCESS. Tube hydroforming process is highly accepted forming process in automobile industry because of its several advantages over conventional manufacturing process such as weight reduction and lower tooling, part consolidations and energy absorbent.

Tube hydroforming is process of producing hollow parts with different cross-section by applying internal hydraulic pressure and axial compressive feed to force tubular blank to confirm to the shape of die cavity. In tube hydroforming there are two major practices: high pressure and low pressure. With the high pressure process the tube is fully enclosed in a die prior to pressurization of the tube. In low pressure the tube is slightly pressurized to a fixed volume during the closing of the die.

In tube hydroforming process, pressure is applied to the inside of a tube that is held by dies with the desired cross sections and forms. When the dies are closed, the tube ends are sealed by axial punches and the tube is filled with hydraulic fluid. The internal pressure can go up to a few thousands of bars and it causes the tube to calibrate against the dies. The fluid is injected into the tube through one of the two axial punches. Axial punches are movable and their action is required to provide axial compression and to feed material towards the center of the tube.

II. PROBLEM DEFINITION

The older approach of stamping and welding together metals for tubing and frames offer a strong final product, but cost and weight of that product were much higher. Not only was this process time consuming and expensive but deficiency like welded joint could be a source of weakness in design but tube hydroforming eliminated this by creating joint from a solid piece of tube.

So, In this study we are going to expanding the branch of Tube 1 as shown in figures, such that it make a joint with Tube 2,while restricting excessive thinning of tube 1.

III. OBJECTIVES

1) The main aim of this study is join the two pipes of same material by hydroforming process and carry out computer simulation for the same for development of tool.
2) Validation of model proposed by comparing its predication with experiment investigation.
3) To study the effect of different parameters such as pressure, axial feed, friction, bulge height on formability of two pipes.
4) To study the formability property of aluminum and copper material in hydroforming process.
IV. DETERMINATION OF MECHANICAL PROPERTY

A. Strain Hardening Exponent (n-value):
The n value, or strain-hardening coefficient, is determined by the dependence of the flow (yield) stress on the level of strain. In materials with a high n value, the flow stress increases rapidly with strain. This tends to distribute further strain to regions of lower strain and flow stress. A high n value is also an indication of good formability in a stretching operation.

B. Anisotropy Index (R-values):
The r value, or plastic strain ratio, relates to draw ability and is known as the anisotropy factor. This is defined as the ratio of the true width strain to the true thickness strain in the uniform elongation region of a tension test. The r value is a measure of the ability of a material to resist thinning. In drawing, material in the angels stretched in one direction (radially) and compressed in the perpendicular direction (circumferentially).

\[ R_0 = \ln \left( \frac{\Delta w}{w_0} \right) / \ln \left( \frac{\Delta T}{T_0} \right) \]

A high r value indicates a material with good drawing properties. The r value frequently changes with direction in the sheet. In a cylindrical cup drawing operation, this variation leads to a cup with a wall that varies in height, a phenomenon known as earing. It is therefore common to measure the average r value, or average normal anisotropy, and the planar anisotropy.

Normal Anisotropy (R) is given by:
\[ R = R_{00} + R_{45} + R_{90} \div 3 \]

C. Data and Calculations:

<table>
<thead>
<tr>
<th>SR NO</th>
<th>DE G</th>
<th>YEILD STRENGTH</th>
<th>UTS</th>
<th>n</th>
<th>K</th>
<th>R</th>
<th>% ELON GATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>O°</td>
<td>2.440</td>
<td>223</td>
<td>0.22</td>
<td>48</td>
<td>0.853</td>
<td>4.0</td>
</tr>
<tr>
<td>2</td>
<td>45°</td>
<td>2.460</td>
<td>223</td>
<td>0.28</td>
<td>53</td>
<td>0.807</td>
<td>4.0</td>
</tr>
<tr>
<td>3</td>
<td>90°</td>
<td>2.475</td>
<td>224</td>
<td>0.32</td>
<td>56</td>
<td>0.720</td>
<td>4.0</td>
</tr>
</tbody>
</table>

Table 1: Data and Calculation

- Engg strain is given by:
  \[ e = \frac{\Delta L}{L_0} = 0.04 \text{ mm} \]

Where
\( e = \) engineering strain (mm)
\( \Delta L = \) change in length (mm)
\( L_0 = \) initial length (mm)

- Engg stress is given by:
  \[ \sigma = \frac{F}{A_0} = 223.0 \text{ N/mm}^2 \]
  where
  \( F = \) load in (N)
  \( A_0 = \) cross section area of the sample (mm²)

- True strain is given by:
  \[ \varepsilon_1 = \ln(1 + e) = 0.039 \text{ mm} \]

- True stress is given by:
  \[ \sigma_1 = \sigma_{uts} (1 + e) = 231.94 \text{ N/mm}^2 \]

D. Theoretical Calculation:
1) Corner Radius of The Die
   \( R_0 = 3t = 4 \text{ mm} \)
2) Internal Pressure Limits
   Maximum internal pressure can be calculated by
   \[ p_{ty} = \sigma_y \frac{2t_o}{D_0 - t_o} - 16 \text{ mpa} \]
   Where,
   \( \sigma_y = \) Yield strength of material
   \( D_0 = \) Outer diameter of tube
   \( t_o = \) Thickness of tube
   3) Sealing force:
   \[ F_{sealing} = \pi \sigma_y t_o R_0 = 3.14 \times 223 \times 0.6 \times 8 = 3361 \text{ mpa} \]

E. DIE Design:

Fig. 3:
Table 2:

<table>
<thead>
<tr>
<th>SN</th>
<th>COMPONENT</th>
<th>L (mm)</th>
<th>W (mm)</th>
<th>T (mm)</th>
<th>D (mm)</th>
<th>R (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Al TUBE</td>
<td>60</td>
<td>-</td>
<td>0.6</td>
<td>16</td>
<td>-</td>
</tr>
<tr>
<td>2</td>
<td>UPPER DIE</td>
<td>210</td>
<td>120</td>
<td>40</td>
<td>-</td>
<td>4</td>
</tr>
<tr>
<td>3</td>
<td>LOWER DIE</td>
<td>210</td>
<td>120</td>
<td>40</td>
<td>-</td>
<td>4</td>
</tr>
</tbody>
</table>

F. Results:

1) FE Simulations Results:

At the beginning of the T-shape hydroforming study as a two crucial parameters the internal pressure and axial stroke were determined. Numerical simulation was performed for different pressure. Increasing the forming pressure causes thinning at the top of the branch.

Fig. 4: Percentage Thickness Reduction Prediction In Al Tube With FE Simulations

Figure 4.1 shows % thickness reduction of the t-shape predicted with FEM for various pressure. The simulation results predicted a sound T-shape with the maximum thinning at the top of the branch that were 32.03%.

Fig. 5: Thinning Prediction In Al Tube With FE Simulations

Fig. 6: Maximum Stress Generated Prediction with FE Simulations

Figure 4.3 shows stress generated in Tube at different pressure. In this area maximum value of the stress was for pressure is 549.03, which is required.

Fig. 7: Expansion of Tube Prediction with FE Simulations

Fig. 8: Forming Limit Diagram

Figure 4.4 shows the possible expansion of tube and figure 4.5 shows the FLD for the Thickness reduction of tube.

V. CONCLUSION

FEA simulation is carried out to find the Effect of geometric parameters on the formability of two pipes. And according to the FE analysis result Experimental set up for joining of two tubes is designed and validation should be carried out. From The Finite element Analysis we conclude that

1) The die corner radius has great influence on formability. An optimum value of 4mm is find out of expansion of branch height without cracking or excessive thinning.

2) Results for thinning and bursting, should be evaluated with care since many other factors such as geometry, material would affect results greatly.

3) Combined geometric parameters has been worked out with FE analysis

VI. FUTURE SCOPE

There is still much to do with the improvement of these and applying them into complex parts. A computer program which divides the forming into various segments and time intervals can be developed to predict all forces, pressure and thinning (or thickening) as Pressure increase.

REFERENCES


