A Review Modeling and Analysis of Crack in the Sheet Metal Forming Process
Sharad Haridas Dethe¹ Dr. Mangesh Phate² Prof. Sachin Jadhav³
¹ME Student (Design Engineering) ²,³Department of Mechanical Engineering
¹,²,³PVPIT Bavdhan, Savitribai Phule Pune University, India

Abstract—All sheet metal forming processes occur after permanent plastic deformation resulting in the material properties variation and therefore it is necessary to determine the extent to which a material can further be deformed for subsequent forming. Normally tentative trial and error procedure was employed to determine these changes and adjust process settings (dies design parameter, loadings, tool course) accordingly. This approach is time consuming and depends heavily on the experimentation and trial. To concentrate on these limitations, sheet metal forming simulations has been applied to detect crack in sheet metal forming process. In this dissertation simulation is carried out on Ansys and Hyper form software, simulation result validation is carried out by experimentation process. By using sheet metal simulation, we can easily analyse the meshing conditions, stress analysis & failure point in sheet metal. After analysing the data, we can easily finalize the forming process parameter & experimentation which is the motive of this dissertation.

Key words: Crack detection, deep draw analysis, deep draw modeling, simulation & validation

I. INTRODUCTION
Drawing is a forming process in which the metal is stretched over a form. Deep drawing typically faces difficulties or challenges associated with wrinkling, thinning, Crack and optimization of shapes. All of which are expensive as they lead to wastage of material and loss of production time. Simulation can predict such defects during product development and often prevent their occurrence during production with the attendant saving time and material, by identifying necessary and often times simple changes in design. It is of importance for sheet metal fabricating companies to adopt product development simulation techniques in order to reduce or eliminate the aforementioned problems and thus remain competitive.

Sheet metal forming is one of the most widely used manufacturing processes in industry that is used to change the geometry of sheet metal. Sheet metal forming is used to produce various products from mild steel, stainless steel, copper, aluminum, gold, platinum, tin, nickel, brass and titanium. To reduce costs and increase the performance of manufactured products, more and more lightweight and high strength materials have been used as a substitute to the conventional steel. These materials usually have limited formability, thus, a thorough understanding of deformation processes and the factors limiting the forming of sound parts is important, from both engineering and economic viewpoints [1].

II. LITERATURE REVIEW
Recently, simulations in the automobile industry are as a rule restricted to the sheet metal forming process, which is where the greatest changes in shape of material. Simulation used today is the result of a development process through the last two decades A main focus of this process was the continuous optimization to characterize the material properties [1]

Single point incremental forming (SPIF) is a new innovative and feasible solution for the rapid prototyping and the manufacturing of small batch sheet parts. The process is carried out at room temperature and requires a CNC machining centre, a spherical headed tool and a simple support to fix the sheet being formed. In incremental Sheet Metal Forming the blank is incrementally deformed in to a desirable shape by hemispherical or ball nose tool travelling along a prescribed path [2].

This paper explore about simulation method in Ansys & Hyper form software. And simulation result validation is carried out for crack in sheet metal forming by experimentation process. By using sheet metal simulation the meshing conditions, stress analysis & failure point in sheet metal easily analyzed.

III. METHODOLOGY
The importance of sheet metal working process in present technology is due to the ease with which metal may be formed into useful shapes. Deep drawing can be done in single draw or double draw [2, 3, 4]. When it is completed in double draw then, it is called re-drawing or restrike operation. Number of draws can be calculated by taking ratio of blank diameter to the required diameter of the cup which is known as Draw Ratio (DR). In this paper three different blanks are chosen with initial thickness of 1.00mm, 1.20mm & 1.50mm with blank outer diameter of 65mm [4].

A. Flow Chart and Methodology:
Methodology of this simulation work starts from the 3D CAD model generation in ANSYS. This CAD model is used further for meshing, analysis & for simulation purpose in HYPER FORM. 3D Cad model can be generated by the majority of approaches used to date still follow the traditional CAD route by using an intermediary step of surface reconstruction which is then followed by a traditional CAD-based meshing algorithm. CAD-based approaches use the scan data to define the surface of the domain and then create elements within this defined boundary [3, 8, 10].

In this paper we have circular CAD model (called as Retainer or holder in industrial definition). The FEA consists of three steps: Pre-processing, Processing (or Solution) and Post-processing. A complete finite element analysis is a logical interaction of these three steps [4, 6, 8, 14].
B. Pre-Processor In Meshing

Pre-processing is the primary step in FEM. It includes geometry creation, meshing and applying boundary conditions. The software ANSYS or HYPERMESH includes tools to create geometry in itself, but it is preferred to create 3D model in CAD software. Then model is to be imported in the analysis software to create meshing. Meshing is the process of discretization of the geometry into elements and nodes. After meshing the suitable boundary conditions are to be applied like Pressure, force, displacement, fixed support etc. In general we can say that pre-processing is the step of creating a deck for the analysis [15].

C. Draw Operation Calculations by Conducting Dynamic Analysis Using Solver:

a) Type Of Operation:

In sheet metal forming operations, the amount of useful deformation is limited by the occurrence of unstable deformation which mainly takes the form of localized necking or wrinkling. Failure by wrinkling occurs when the dominant stresses are compressive, tending to cause thickening of the material. Localized necking occurs when the stress state leads to an increase in the surface area of the sheet at the expense of a reduction in the thickness. The two kinds of neck that occur are diffuse necking (so called because its extension is much greater than the sheet thickness), and the localized necking (through thickness thinning), which is terminated by final separation or fracture. After the localized neck initiates, further deformation of the material concentrates in this localized region, and homogeneous deformation away from neck region vanishes completely [5]. The localized neck is therefore a very important phenomenon in determining the amount of useful deformation that can be imposed on a work piece. The mechanism for initiation of localized band involves a number of factors including material properties and punch profile. The phenomenon is attributed to the softening effect, including geometric softening (the decrease with strain of the cross-section area which bears the forming load, the generation of voids), and material softening (flow stress decreases with the increase of the effective strain).

\[
\frac{h}{d} \leq 0.5 \quad \text{shallow drawing} \quad \ldots \quad (1.1)
\]

\[
\frac{h}{d} > 0.5 \quad \text{deep drawing}
\]

Where, \( h/d = 23.50/45.4 = 0.6 \)  
(Deep drawing operation)

h = shell height

\( d \) = shell diameter
b) Estimation Of Blank Diameter (Theoretical):

\[ D = \sqrt{(d^2 + 4dh - 0.5r)} \quad \ldots \text{(1.2)} \]
\[ = \sqrt{(45.4^2 + 445.4^2 + 17.00 - 0.5*3.00)} \]
\[ = \sqrt{(45.4^2 + 445.4^2 + 17.00 - 0.5*3.00)} \]
\[ = 65.00\text{mm} \quad \text{(Blank diameter)} \]

Where,
- \( D \) – Blank diameter in mm
- \( d \) – Shell outer diameter in mm
- \( h \) – Shell Height in mm
- \( r \) – Corner radius of punch

When \( d / r \) is between 15 & 20

\[ \text{Trim allowance} = 0.005\text{mm for every 10 mm diameter of drawn cup} \quad \ldots \text{(1.3)} \]

Where,
\[ D_1 = \text{Initial diameter of blank (D1)} \]
\[ = D \text{ (Theoretical diameter.)} + \text{Trim allowance} \]
\[ = 65.00 + 0.005*45.40 = 65.224\text{mm} \]

\[ T / D \text{ Consideration: Following Results Of Wrinkling Are Obtained From Sheet Thickness To Blank Diameter Ratio On The Basis Of Simulation In Hyper Mesh.} \]

<table>
<thead>
<tr>
<th>( T / D ) (%)</th>
<th>Severity of wrinkling</th>
</tr>
</thead>
<tbody>
<tr>
<td>Up to 0.5</td>
<td>Wrinkling is a severe and compressive load must be reduced. Blank holder must be used, so a double action press is preferable</td>
</tr>
<tr>
<td>Above 0.5 up to 1.5</td>
<td>Wrinkling is moderate and low. Blank holding forces are permitted</td>
</tr>
<tr>
<td>Above 1.5 up to 2.5</td>
<td>Wrinkling is very light so, single action press is enough</td>
</tr>
</tbody>
</table>

Table 1: \( T / D \) Decides The Severity Of Wrinkling

IV. SIMULATION, EXPERIMENTATION & VALIDATION REVIEW

A. Simulation:

In sheet metal forming simulation, the forming of sheet metal is simulated on the computer with the help of special software. Simulation makes it possible to detect errors and problems, such as wrinkles or splits in parts, thinning or crack on the computer at an early stage. In this way, it is not necessary to produce real tools to run practical tests. Simulation helps in becoming established in the automotive industry since it is used to develop and optimize every sheet metal part.

To illustrate the metal forming process, there must be a model of the real process. This is calculated in the software using the finite element method based on implicit or explicit incremental techniques. The parameters of the model must describe the real process as accurately as possible so that the results of the simulation are realistic.

B. Experimentation:

will be conducted on a hydraulic press of a suitable capacity. The die would be mounted on the bolster plate of the press and the speed of the ram would be set based on the historical data as well as the input received from the analysis & simulation data. Forming problems can be predicted before tool fabrication through the use of software that can be integrated into production routes which rely increasingly on computer technology. The prediction of forming difficulties at the component design stage ensures that the chosen geometry is compatible with the draw ability of steel. Drawing has become a highly technical process, and the development of a steel forming route no longer involves simple trial and error methods. Close collaboration between component designers, drawing engineers and steelmakers guarantees the industrial feasibility of new parts with very short development times.

- The parameters influencing the draw operation are:
  - Type of material
  - Thickness of the component
  - Mechanical properties, especially the Limiting Draw Ratio
  - Use of lubricant
  - Blank size and development
  - Blank holding pressure
  - Speed of the operation

C. Validation Review:

Validation of this simulation data carried out as, the appropriate capacity press selected by knowing the drawing load. Working with the presses of higher capacities may lead to many types of defects such as cracks and tearing. Blank holder pressure needs to optimize over a given range for optimized geometry. The coefficient of friction needs to be optimized for the new geometry. Generally the deep drawing objects are analyzed for their strength and failures with circle grid analysis, which is practically carried out on a sample piece, which is known as formability analysis. Alternatively, the actual trials performed over the component would directly reflect over the ease of ‘drawing operation’ offered for the said simulation data in Ansys or Hyper form.

V. CONCLUSION

To overcome the entire problem related to thinning & crack in sheet metal forming simulation realized follows;

- Simulation & Finite Element Method is easy to use and user friendly methods in sheet metal forming crack analysis, simulation & optimization.
- Use simulation reduces time as well as production & development (trial & error) cost in sheet metal.
- Through all above references available, simulation appears to be an efficient choice for optimization of the forming process.

VI. PROPOSED FUTURE SCOPE

1) To do simulation of the actual sheet metal component over cad model with crack detection in simulation phase for material thickness up to 1.50mm.
2) To do the physical experimentation & validation of the simulation & analysis results.
3) To do analysis of fracture & wrinkle defect in forming process.

REFERENCES


[4] Sponsorship of this project is given by AESSEAL INDIA PVT LTD, Pune. Problem definition is defined by customer of this sponsored company to resolve the sheet metal thinning & crack issue with the help of modelling, simulation, new die design & development with reference to the simulation in forming operation.

[5] https://www.google.co.in/?gws_rd=cr&ei=uvLLUs2jGcGkrQeOp4CQAg#q=3331+ch7-Design+of+press+working+tools 3331 ch7-Design of press working tools


