Finite element simulation and Experimental verification of Incremental Sheet metal Forming

Krishna Kaushik Yanamundra, Dr. R Karthikeyan, Dr. Vishal Naranje
Mechanical Engineering Department, BITS Pilani, Dubai Campus

Abstract: Incremental sheet metal forming is now a proven manufacturing technique that can be employed to obtain application specific, customized, symmetric or asymmetric shapes that are required by automobile or biomedical industries for specific purposes like car body parts, dental implants or knee implants. Finite element simulation of metal forming process is being performed successfully using explicit dynamics analysis of commercial FE software. The simulation is mainly useful in optimization of the process as well design of the final product.

This paper focuses on simulating the incremental sheet metal forming process in ABAQUS, and validating the results using experimental methods. The shapes generated for testing are of trapezoid, dome and elliptical shapes whose G codes are written and fed into the CNC milling machine with an attached forming tool with a hemispherical bottom. The same pre-generated coordinates are used to simulate a similar machining conditions in ABAQUS and the tool forces, stresses and strains in the workpiece while machining are obtained as the output data. The forces experimentally were recorded using a dynamometer. The experimental and simulated results were then compared and thus conclusions were drawn.

Keywords: Incremental Sheet Forming, tool path planning, finite element simulation, explicit dynamic analysis, CNC, CATIA, ABAQUS.

1. INTRODUCTION:

Sheet Metal forming is a very familiar process and rather conventional the way it involves dies and punches. This process demands geometrically active dies, which may be cost effective for large scale production but for a small scale prototyping it can be expensive taking all the material costs into consideration. In order to deal with this Incremental sheet metal forming can be employed.

Incremental sheet metal forming (hereby referred as ISF), is a stretch forming process which does not involve a female die, it simply has 3 key elements, a Holder to hold the plate encastered at its periphery, a modified tooling used specifically for this process which has a hemispherical bottom, a CNC milling machine. G codes for the required shapes are written and fed into the CNC, then the ISF tool is attached, the workpiece is placed in the holder and the program is run. The key step in this process is to plan the tool path. The tool travels and causes a localized deformation in the sheet and such multiple deformations result in the final shape. Using this method even the most complex shapes can be produced with a relatively low energy demand and tooling cost. [1]
2. METHODOLOGY

The procedure followed in this particular experiment can be broadly split into 3 stages which are explained in detail below.

*Generation of G codes.*

In this particular stage first the solid geometry of the required shape is modelled using a commercial 3D modelling software called CATIA. Once the model is ready the G code and the coordinate points can be generated using the surface machining module available in the software.

*Finite Element Analysis*

This is where the usage of ABAQUS - a FEM software is incorporated, the steps followed in order to simulate the process is as illustrated in the flowchart Fig 2.1.

Fig 2.1: Flow of steps to be followed while simulation the model in ABAQUS
Creating the Solid models

In this phase a Standard/Explicit model has been created with 2 parts, the sheet from which the shape is supposed to be formed out of and the tool that deforms the sheet. The type of the part must be chosen in accordance with its nature. Thus for Part_1 (Alu_Sheet), the part is chosen to be a 3 dimensional, deformable, planar shell rectangular in shape with dimensions 165 X 150 mm. Part_2 (Tool) will be 3D, Analytical rigid.

Mesh

In order to perform any numerical simulation using finite element method meshing plays a key role in determining the accuracy of the solutions for the problem that is being solved. In order to maintain the precision of the results the mesh must be fine and refined. In this case only the sheet needs to be meshed as the tool is an analytical rigid part. The sheet will be meshed with shell elements with a certain shell thickness. Before meshing the part the edges must be seeded in order to control the cell size followed by which the mesh controls are assigned where the element type is set to be Triangular (S3R) for simplicity while problem solving.
Creating the Material models

In this section the material needs to be defined that we want to assign only to the Alu_sheet, i.e. the Physical and mechanical properties of Aluminium – Mass density, Elasticity (isotropic) young’s modulus and poisons ratio, Plasticity. Plasticity is defined according to the coordinates extrapolated from the stress strain curve that is obtained after performing the tensile and compression tests in the UTM. The material property data thus obtained is as mentioned below in Table 2.2 and Fig 2.5

Table 2.1: Elastic Material Properties

| Property            | Value   |
|---------------------|---------|
| Young’s Modulus     | 70000 Mpa |
| Poisson’s ratio     | 0.33    |

Fig 2.5: Stress – Strain curve obtained from the testing performed in UTM
Table 2.2: Stress – Strain values (Test observations)

| Stress | Plastic strain | Plastic strain | Plastic strain | Plastic strain | Plastic strain | Plastic strain |
|--------|----------------|----------------|----------------|----------------|----------------|----------------|
| 44.8   | 0              | 159.5          | 0.75           | 176.92         | 1.7            | 185.30         | 2.5            | 191.59         | 3.3            |
| 95.5   | 0.01           | 166            | 1              | 178.13         | 1.8            | 186.18         | 2.6            | 192.27         | 3.4            |
| 126    | 0.1            | 167.90         | 1.1            | 179.29         | 1.9            | 187.02         | 2.7            | 192.94         | 3.5            |
| 132    | 0.15           | 169.67         | 1.2            | 180.40         | 2              | 187.84         | 2.8            | 193.6          | 3.6            |
| 137    | 0.2            | 171.31         | 1.3            | 181.46         | 2.1            | 188.64         | 2.9            | 194.24         | 3.7            |
| 143.5  | 0.3            | 172.84         | 1.4            | 182.48         | 2.2            | 189.41         | 3              | 194.86         | 3.8            |
| 148.5  | 0.4            | 174.28         | 1.5            | 183.46         | 2.3            | 190.15         | 3.1            | 195.47         | 3.9            |
| 153    | 0.5            | 175.63         | 1.6            | 184.4          | 2.4            | 190.88         | 3.2            | 196.06         | 4              |

After the material model is created a section of the alu_sheet must be assigned with a homogeneous material of the previously defined properties.

**Assembly**

In this particular section the instances of the tool and the sheet must be created and the reference point at the bottom center of the tool must be translated to the center of the sheet which is the origin.

![Fig 2.6: Tool sheet assembly with tool at origin](image)

**Steps**

The analysis is divided in to 3 steps namely – Initial, Goto_touch and Deform. Initial step is where the interaction between the tool and the sheet and the contact controls are defined these
properties are propagated into the further steps in the simulation. Boundary conditions are defined in individual steps.

**Boundary conditions:**

The boundary conditions are defined within the steps and few of which are propagated and the rest are modified from step to step. In the initial step only one boundary condition is defined i.e. fixing the edges of the edges of the sheet using the encastre command which will make \( u_1, u_2, u_3 \) which are the displacements in all the 3 directions and \( ur_1, ur_2 \) and \( ur_3 \) which are the rotations about \( x, y \) and \( z \) axes of the cells along periphery of the sheet zero. This boundary condition (BC_1) is propagated to the rest of the steps without any modifications.

The second boundary condition is assigned in the second step i.e. Goto_Touchdown where \( u_1 \) and \( u_2 \) values are given to be the initial \( x \) and \( y \) coordinates of the tool to the reference point of the rigid tool. This makes the tool move from the origin or center of the sheet to the starting point of the tool path.

In the third and final step the second boundary condition is modified to only \( u_1 = 1 \) with an amplitude \( Amp_x \) assigned to it. \( Amp_x \) defined the tool path in \( x \) direction with respect to time steps. Similarly two more boundary conditions are defined with \( u_2 = 1 \) with amplitude \( Amp_y \) and \( u_3 = 1 \) with amplitude \( Amp_z \). In the final boundary condition along with \( z \) motion definition and have also set \( ur_1, ur_2 \) and \( ur_3 \) of the tool to be zero.

These boundary conditions help in defining the tool path during the simulation. Tool path can be defined for the same geometry in two different ways, one is called helical where the path is continuous with all three \( x, y \) and \( z \) coordinates changing monotonically with time. Other way to define the tool path is steps i.e. one set of coordinates in the \( xy \) plane followed by a decrement in \( z \) direction and then again another set of coordinates.

![Fig 2.7](image) Tool path definition (a) helical or spiral, (b) Step wise with draw angle alpha.

**Interaction Properties:**

Interaction between the bottom surface of the tool and the top surface of the sheet must be defined in order to simulate the behavior of the sheet under deformation caused by the rigid tool. This is defined using a Master slave system. Master is the rigid surface that deforms which in this case is the bottom surface of the tool and the surface that follows the master and deforms will be the slave surface which in this case is the top surface of the sheet. Define the general contact between these two surfaces with a certain coefficient of friction at default conditions. Also have to link this particular interaction property with a contact control.
Output requests

Before submitting the job to assign the history output requests and field output requests is necessary. In the field output requests there are PE – plastic strain components, PEEQ – equivalent plastic strain, S – Stress components and invariants, STH – Section thickness, U – Translations and rotations, UR – rotations. And for the history output requests there are all the totals of energy components - ALLAE, ALLCD, ALLDMD, ALLEE, ALLFD, ALLIE, ALLJD, ALLKE, ALLKL, ALLPD, ALLQB, ALLSD, ALLSE, ALLVD, ALLWK, ETOTAL.

 Experimental Validation

The experimental setup is as illustrated in the fig. there is a frame which supports the sheet by fixing it about its periphery and the tool is replaced by a hemispherical bottomed forming tool in a CNC. G codes are fed and run. Grids are made on the sheet using a gridding machine. The size of the grids are compared before and after forming in order to examine the local strain.

3 Results and Discussions

The simulation is run and the results are obtained. The tool motion in the y and z directions are plotted from the coordinate points against the times steps. The plots are illustrated in fig 3.1 and fig 3.2 below.
Post analysis plots are also obtained for the stresses and the energy over the time period of the simulation of all the elements in the sheet. The stresses in the system intuitively increase as the depth or the tool motion in the z direction increases. The contour plots are plotted on the surface of the deformed sheet and is shown in Fig 3.3. The plot depicts the mises stresses on the sheet.
Fig 3.4 depicts the final deformed sheet. This particular resultant deformed sheet is compared to the experimentally formed aluminium sheet of the same shape as simulated in ABAQUS. The Fig 3.5
depicts the actual undeformed sheet to the deformed sheet at the end of the simulation. The variations from the actuals are noted.

Fig 3.1: Experimentally deformed sheet top view (left) and bottom (right) vs. Simulated results.
Fig 3.2 Tool path in Z direction.

Fig 3.3 Tool path in Y direction
Fig 3.4 Energy v/s time graph

Fig 3.5 Stress v/s Time graph
4 Applications and Future scope

This particular forming technique can be implemented in cases with small scale prototyping and with specific applications such as knee and dental implants. These shapes vary from person to person so doing it the conventional way by making a mould of every different sample every time is extremely time consuming and demands accuracy. Instead employing this method makes the job much easier.

Future studies in this field include heat dissipation due to friction while forming, spring back reduction as it is a hurdle in achieving accuracy. Also incremental sheet forming in the case of polymers and composites is a field of great importance. On the other hand tool wear, surface roughness and machining time can be improved.

5 References

[1] Guptaa P, Jeswieta J – “Observations on heat generated in single point incremental forming”, 17th International Conference on Sheet Metal, SHEMET17.

[2] William L E, Grimm T J, Ragai I and Roth J T – “Optimum Process Parameters for Springback Reduction of Single Point Incrementally Formed Polycarbonate”, 45th SME North American Manufacturing Research Conference, NAMRC 45, LA, USA.

[3] Mohanty S, Regalla S P, Rao Y V D – “Tool Path Planning for manufacturing of Asymmetric parts by incremental sheet metal forming by means of Robotic manipulator”, 5th International Conference of Materials Processing and Characterization (ICMPC2016).

[4] Wang J, Nair M and Zhang Y – “An Efficient Force Prediction Strategy in Single Point Incremental Sheet Forming”, 44th Proceedings of the North American Manufacturing Research Institution of SME.

[5] Yamashita M, Gotoh M, Atsumi S – “Numerical simulation of incremental forming of sheet metal”, journal of materials processing technology 199 (2008) 163–172.
[6] Ben Said L, Mars J, Wali M, Dammak F, Numerical prediction of the ductile damage in single point incremental forming process, *International Journal of Mechanical Sciences* (2017).

[7] Azevedo N G, Farias J S, Bastos R P, et al., 2015. Lubrication aspects during single point incremental forming for steel and aluminum materials. *International Journal of Precision Engineering and Manufacturing* 16 (3), 589–595.

[8] Bambach M, Hirt G, 2007. Error analysis in explicit finite element analysis of incremental sheet forming. *AIP Conference Proceedings*, Aip 908 (2007), 859–864.

[9] Bastos R P, Alves de Sousa R J, Ferreira J A F, 2015. Enhancing time efficiency on single point incremental forming processes. *International Journal of Material Forming*. 