# Simulation and Experimental Evaluation of Multi Point Incremental Forming On Titanium Grade 2 Sheet

**Arun A S[1] Dr. C. Raju[2] K A Selvarajan[3]**
[1,2]Department of Mechanical Engineering [3]Department of Production Engineering
[1,2]College of Engineering Trivandrum, Kerala, India [3]National Institute of Technology, Trichy
Tamilnadu, India

*Abstract—* Incremental forming has great potential applications since it can be used in small batch production process and rapid prototyping applications. There are many advantageous effects when we implementing incremental forming over conventional forming techniques. Due to the implementation of CNC human effort can be reduced and also the process can be easily automated. Incremental forming does not use any dedicated die for the manufacturing of desired shapes. Here in this paper detailed explanation of how the strain become predicted when we carry out multi point incremental forming process by Abaqus explicit solver and it became compared with the experimental result. For the experimental process MPIF tool is used which has 6 number of balls and is used to decrease the processing time and improving accuracy.
*Key words:* MPIF, CNC, Abaqus

## I. INTRODUCTION

Incremental forming can be generally classified into three types. First one is single point incremental forming. There are many research works are going on in the field of simulation of single point incremental forming process. Chennakesava reddy et al. in his journal conduct numerical simulation of single point incremental forming and also validate the result with experimental results. He conduct the forming process in phosphor bronze sheet and obtain the result that FLD (forming limit diagram) was highly influenced by the sheet thickness [1].

Krishna koushik yanamundra et al. conduct numerical simulation of single point incremental forming on aluminium sheet and describes the procedure for find out the experimental and numerical strain. Also he explain how to extract the coordinate points from the stress strain curve inorder to feed the value to the software [2].

Kurra suresh et al. in his paper describes how to generate the tool path for various simulation software such as abaqus, Ls dyna, anasys etc. He suggest the method of generation of coordinate points arithmetically [3]. The second type of incremental forming is two point incremental forming process. In this process a partial die is used on the lower side of the plate.
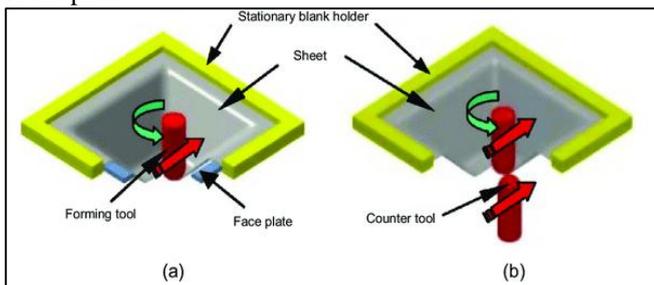


Fig. 1: (a) single point IF (b) Two point IF[5]

Baddredin saidi et al. in his paper describes how to optimize single point incremental forming on titanium sheet by combining simulation technique and also response surface methodology. He develop a second order quadratic equation in order to predict the forming forces [4]. The next type of incremental forming is that multi point incremental forming. Multi point incremental forming is carried out by either altering the configuration of the tool or by altering the configuration of the fixture. Here in this research multi point incremental forming process is carried out by altering the configuration of the tool. Some authors in the field of multi point incremental forming claims that it is hybrid forming process and it give the advantages of both the single point incremental forming and two point incremental forming. Multi point incremental forming can be also formulated by changing the die in two point incremental forming [6].

## II. EXPERIMENTAL METHODOLOGY

### A. Multi point incremental forming tool

Mainly multi point incremental forming tool has 3 parts.
1. A shank with threaded end.
2. Mild steel cap with 45 degree inclined hole in it.
3. Hardened steel ball.



Fig. 2: Multi point incremental forming tool (patent protected)

The main advantage of using this tool in incremental forming process is that forming time can be reduced and also accuracy can be improved. One of the important advantage is that by using the multipoint incremental forming tool spring back effect can be considerably reduced.

### B. Multi point incremental forming process

For conducting multi point incremental forming process a vertical CNC milling machine is used. G code is fed into the

CNC machine in order to form a conical frustum on the Titanium grade 2 sheet and the sheet is clamped by using a fixture which is specially made for the incremental forming process.


Fig. 3: Multi point incremental forming process.

In order to measure the experimental strain, circular grid is made on the lower side of the sheet. The grid is made on the sheet is by laser engraving technique. By measuring the deformed circular dimensions such as major circle diameter and minor circle diameter we can find out the total strain.


Fig. 4: Circular grid is made on the lower side of titanium grade 2 sheet.

The experiment is continued till 36 minute. That is 36 minute is required for deforming the sheet to 20mm depth. After reaching 20mm depth the deformed sheet is unclamped. The deformed shape is shown in fig5.


Fig. 5: Conical frustum shape is formed on the sheet.

The major circle diameter and minor circle diameter are measured by travelling electron microscope. The major true strain and minor true strain values are obtained from the equations explained in modified cockroft-Latham criterion. Forming limit diagram can be drawn by taking the values of major circle diameter and minor circle diameter and which gives the idea that how extent the material deform by multi point incremental forming process.

Engineering stress vs strain graph was plotted by conducting tensile test on UTM machine. These values were converted into true stress and true strain values using the equation explained by Raju et al.[8]

## III. NUMERICAL SIMULATION

### A. Generation of CNC codes

In this Particular stage first the solid geometry of the required shape is modelled using a commercial 3D modelling software called PRO E. Once the model is ready the CNC code and the coordinate points can be generated using the manufacturing module available in the software.
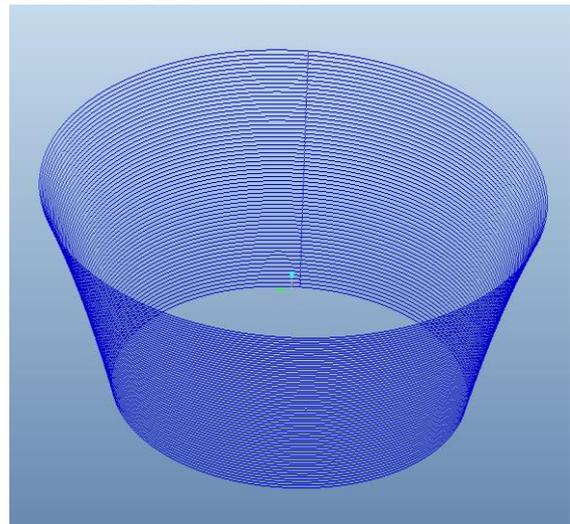

Fig. 6: Tool path generate by pro e software

The CNC code obtained from the software is directly fed into the vertical CNC milling machine. This CNC cade can be also used for finding the coordinate points required for defining the tool motion on the sheet in simulation process. The X, Y and Z coordinates are required for giving feed rate in the Abaqus software.

## B. Creating the Solid models

Thus for Titanium sheet the part is chosen to be a 3 dimensional, deformable, planar shell rectangular in shape with dimensions 150 X 150 X 0.5 mm.
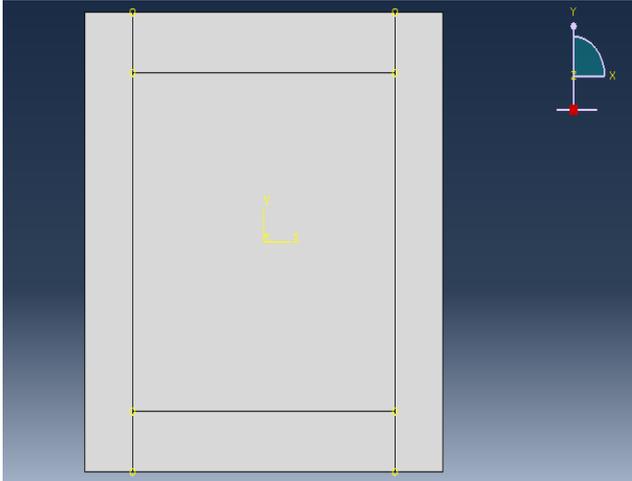


Fig. 7: Solid model of the sheet.

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.
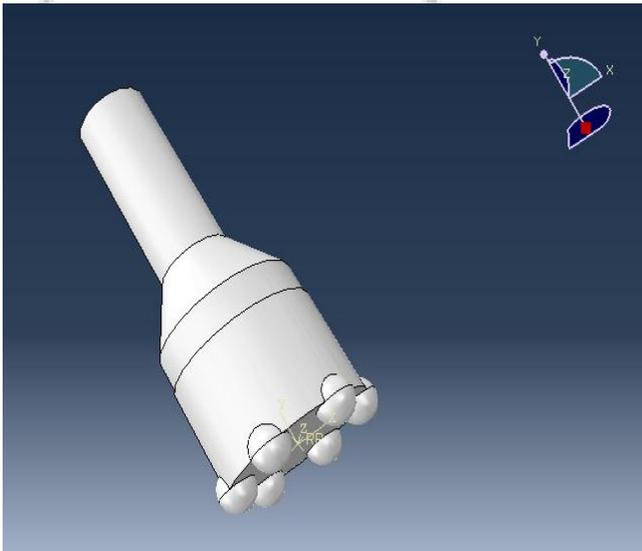


Fig.8: Solid model of MPIF tool

For the Titanium grade 2 sheet the material is chosen to be 3D deformable planar structure and for the multipoint incremental forming tool become 3D discrete rigid structure.

## C. Meshing

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 the sheet and the tool needs to be meshed because both are discrete parts.
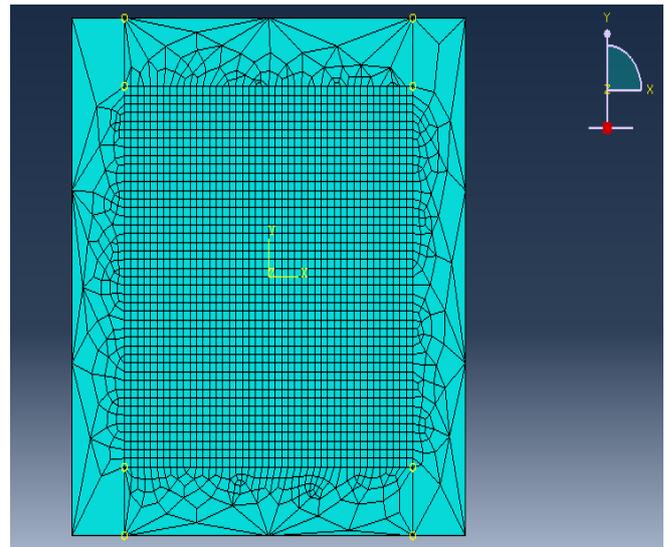


Fig. 9: Meshing of Titanium grade 2 plate.

The mesh size is 2mm and the element type chosen here is S4R which is a 4 node quadrilateral node and used here in order to reduce the computation. Same node and element size is used for multipoint incremental forming tool.

## D. Boundary conditions

One of the main process in the incremental forming simulation is proper definition of the boundary condition. The first process is fixing all the four sides of the plate by encastre command (fig10). That is by using the above mentioned command the values of u1, u2, u3 values given as zero. Then next process is defining the motion of the tool. The process of giving the motion of the tool is more complicated session in the simulation process. For the X direction movement, the X coordinate obtained from the Pro e software is given to the software monotonically with time in amplitude module. Amplitude module also serve as the feed rate of the given process. Thus for Y coordinate and Z coordinate, the above described process is carried out.
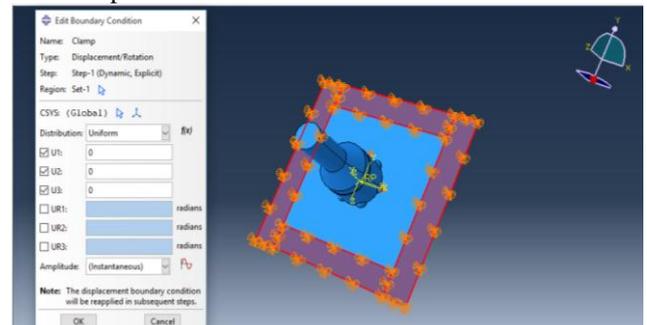


Fig. 10: Boundary condition for fixing the four sides of the sheet.

In order to give the rotation component select the velocity acceleration module available in the software. In the experiment we use 450rpm as speed. Convert this rpm to radians per second and fed to the velocity module.

## IV. RESULT AND DISCUSSIONS

### A. *Finding out The Maximum Stress Obtained from the Simulation Result*
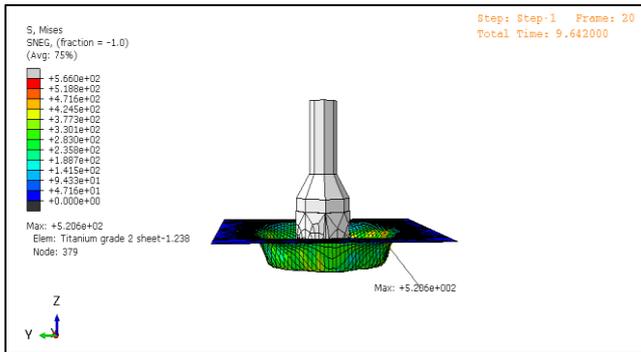


Fig. 11: Maximum stress on the side walls of the cup

For the measurement of stress and strain from experimental method, first process is the formation of circular grid on the back side of the plate. The formation grid is made by the help of laser engraving technique. The grid size is 2mm and it is made on the noncontact surface of the sheet and multi point tool. The computation time for the Abaqus explicit solver becomes 30 hour. The incremental forming process actually completed in 35 minutes. The maximum stress that obtained in the sheet metal is 520.612Mpa and it is located on the side walls of the cup. The minimum stress that obtained in the simulation process is located on the base surface of the plate. This is because the multipoint tool does not have the path to deform the base part of the sheet. The combined effect of stress and strain occurred in the forming process cause the deformation of the sheet.

### B. *Comparing the Strain Obtained From the Simulation and Experimental Methodology*

The strains obtained from the experiment are in good agreement with the strains obtained from simulation. The engineering strains obtained from the simulation for 0.5 mm thick and 20 mm deep cup is 2.171. Whereas the experimental engineering strain of 0.5mm thick conical frustum become 2.1638. The experimental strain is measured by grid analysis with the help of travelling microscope.
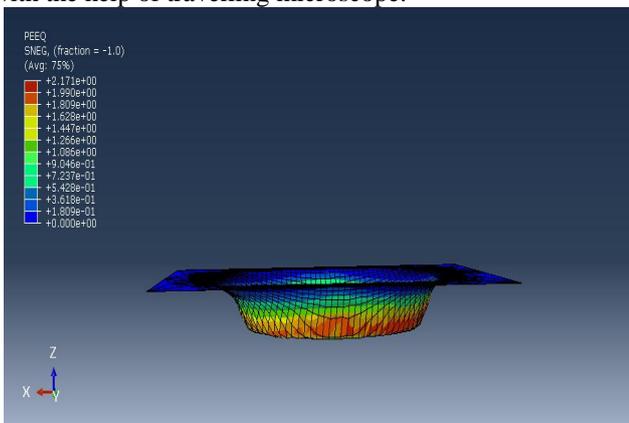


Fig. 12: Engineering strain obtained from the simulation

When comparing simulation strains with the experimental strains, the strains obtained from the experiment are found to be on the lower side. This is because of absence of rupture in the experimentally formed cups. The strains obtained from simulation are greater because they represent the maximum

values of rupture. This further indicates experimental strains are within the acceptable limits of the formability of cup. It is the side walls of the cups which experienced the greatest reduction in thickness. This is due to the reason that the side wall is the most strained part in the formed component. So, the elements in this part of the cup are more strained when compare to other elements. The thickness reduction in the flange and the bottom of the cup was insignificant.

### C. *Comparing the Stress Pattern Obtained from the Simulation Result and Experimental Result.*

To validate the simulation results, the finite element grid of 2mm size was created on the backside of the cup material. The size of element was also 2mm in case of simulation results. The stress pattern obtained by the finite element analysis coincides with the pattern on the cups.
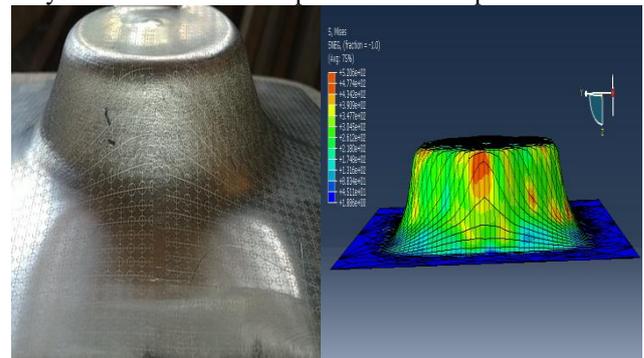


Fig. 13: Deformation pattern obatained from both simulation and experimental result

It is clearly noticed that the deformation pattern obtained from the analysis and simulation methods correctly matched. We can also notice the clear bending and elongation of lines that is made in the actual titanium grade 2 sheet by the help of laser engraving techniques. In order to validate the experimental result the mesh size of 2mm is selected in Abaqus software and after the simulation result 99 percent visual similarity obtained in the stress pattern.

The equivalent stress increases from one contour to the next until it reaches a first local maximum around circumferential region. Then it decreases slightly and increases again to reach its global maximum around lower side near the fracture point. As a reminder, both the circumferential and the radial strains have their maximum around the region where the rupture occurs. However, there is no steady state region at the end of the process, as was the case for the thickness, the tool force, and the circumferential, radial, and in plane shear strains.

## V. CONCLUSIONS

Simulation of multi point incremental forming is successfully carried out in Abaqus explicit solver. A dynamic analysis is carried out in order to find the maximum stress developed on the sheet and also find out the location where it affects. The result is that maximum 520.612Mpa stress is developed after multi point incremental forming and which is located on the side walls of the cup.

One of the important evaluation is that predicting the strain occurred after the forming process. Prediction of strain is more important because it give the idea, how extent the multipoint incremental forming can be carried out. For 20mm

deep cup the maximum strain occurred is 2.1638 which is obtained from experimental methodology and the output of simulation gives the satisfied value as the experimental result

### REFERENCES

[1] Chennakesava Reddy, 'Numerical and Experimental Investigation of Single Point Incremental Forming Process for Phosphorus Bronze Hemispherical Cups'. International Journal of Scientific & Engineering Research, Volume 8, Issue 1, January-2017.

[2] Krishna Kaushik Yanamundra, Dr. R Karthikeyan, Dr. Vishal Naranje, 'Finite element simulation and Experimental verification of Incremental Sheet metal Forming'. IOP Conf. Series: Materials Science and Engineering 346 (2018) 012075 doi:10.1088/1757-899X/346/1/012075.

[3] Kurra Suresh, Arman Khan, Srinivasa Prakash Regalla. 'Tool path definition for numerical simulation of single point incremental forming'. Procedia Engineering 64 (2013) 536 – 545.

[4] Badreddine Saidi, Laurence Giraud-Moreau, Abel Cherouat. 'Optimization of the single point incremental forming process for titanium sheets by using response surface'. MATEC Web of Conferences 80,DOI:10.1051/matecconf/20168010011NUMIFORM 2016.

[5] Lang Lihui, Liu Kangning, Gaoshen Cai, Xiying Yang, Chan Guo, and Guoliang Bu. 'A critical review on special forming processes and associated research for lightweight components based on sheet and tube materials'. L. Lihui et al., Published by EDP Sciences, 2014.

[6] J. Naranjo, V. Miguel, A. Martínez-Martínez, L.M. Gómez-López, M.C. Manjabacas, J. Coello. 'Analysis and simulation of Single Point Incremental Forming by ANSYS'. Procedia Engineering 132 (2015) 1104 – 1111

[7] Kurra Suresh, Arman Khan, Srinivasa Prakash Regalla. 'Tool path definition for numerical simulation of single point incremental forming'. Procedia Engineering 64 (2013) 536 – 545.

[8] C. Raju C. Sathiya Narayanan 'FLD and Fractography Analysis of Multiple Sheet Single Point Incremental Forming' The Indian Institute of Metals - IIM 2015.

[9] Petr Homola, Lucie Novakova, Vaclav Kafka, Mariluz P. Oscoz 'Effect of Incremental Forming Parameters on Titanium Alloys Properties' World Academy of Science, Engineering and Technology Vol:8, No:1, 2014

[10] Tyler J. Grimm, Ihab Ragai, and John T. Roth 'A Novel Modification to the Incremental Forming Process', Part1:Multi-Directional Tooling Procedia Manufacturing 10 (2017) 510 – 519.

[11] D. Rees, Basic Engineering Plasticity: 'An Introduction with Engineering and Manufacturing Applications'. Elsevier Science UK (2014), pp. 294-298