

# FE Simulation of Hot Single Point Incremental Forming Assisted by Induction Heating

Amar Al-Obaidi

*Institute for Machine Tools and  
Production Processes (IWP)  
Chemnitz University of Technology  
Chemnitz, Germany  
amar.al-obaidi@mb.tu-chemnitz.de*

Verena Kräusel

*Institute for Machine Tools and  
Production Processes (IWP)  
Chemnitz University of Technology  
Chemnitz, Germany  
verena.kraeusel@mb.tu-chemnitz.de*

Martin Kroll

*Institute for Machine Tools and  
Production Processes (IWP)  
Chemnitz University of Technology  
Chemnitz, Germany  
martin.kroll@mb.tu-chemnitz.de*

**Abstract**— Single Point Incremental Forming (SPIF) of high-strength is challenging. It requires high forming-forces, also the materials exhibit relatively low formability and distinctive springback behavior. Consequently the forming process is limited and the geometrical accuracy is decreased. Therefore hot SPIF was introduced. In the present study, with induction assisted locally heated single point incremental forming (IASPIF) a variant of this approach is investigated. This process was accomplished by forming on the upper side of the sheet using the forming tool while the heat was generated by a circular heating disc moving on the lower side of the sheet. The heat was transferred from the heating disc to the sheet during forming. The sheet was heated locally by this method, in which the heating disc was following the forming tool, by applying the same path for both of them.

**Keywords**— Induction heating, Single point incremental forming, High strength steel, FE modeling

## I. EXPERIMENTAL PROCEDURE

The investigated component was a square pyramid with wall angle of  $62^\circ$  formed from 1.2 mm thick DP1000 steel. The stated sheet metal was selected because it represents a modern high strength steel dual phase steel with a wide range of applications. The modeled sheet was divided into three layers in thickness direction and investigated. Moreover, in the sheet model and a strip of meshes was selected to record the mechanical and thermal behavior of the sheet metal after forming.

## II. THE OBJECTIVE OF THE RESEARCH

The Abaqus CAE program was used to perform finite element simulations of this SPIF process. The simulation process of the IASPIF was carried out to investigate the following effects:

- Effect of heating temperature on forming forces in the IASPIF process
- Calculation of the equivalent effective plastic stresses and strains
- Position and value of the maximum stress and strain that appear during forming
- Thinning behavior during forming in the formed wall part by maximum allowable plastic strain in the thickness direction and comparison with experimental results

## III. FE SIMULATION

Two methods were applied for the numerical simulation of HSPIF: a Plastic behavior model and a fully coupled model.

The plastic properties, represented by the flow curves, of the DP1000 steel sheet metal were calculated from the tensile properties obtained by tensile tests performed at different temperatures ranging from room temperature to  $700^\circ\text{C}$ . Therefore, the Swift law was utilized by considering the hardening exponent. The resulting flow curves were used in the metal's mechanical plastic properties within the Abaqus program model.

The Abaqus program has the ability to analyze the induction heating by coupled simulation between the heat transfer and the electromagnetic process. The simulation of induction heating requires the surfaces of the components inductor, air and the workpiece to be continuously fixed together during simulation. Moreover, the meshing elements must be combined with each other for the three aforementioned components. The problem in simulating the IASPIF process with the Abaqus program is the continuous movement of the tool along a spiral path, resulting in high strains. At the same time, the inductor is moved simultaneously with the tool under the sheet with the same path. In the Abaqus program, it is not possible to simulate the movable tool and inductor simultaneously due to the fact that the four modeled basic parts (air, tool, sheet and inductor) must statistically be combined together during analysis. Therefore, alternative methods were utilized to heat up the sheet during IASPIF instead simulating the induction process. The process was substituted by heating the formed sheet with a heating disk in the lower side of the sheet. Furthermore, the heat generated in the sheet was caused by the effect of heat transfer from the heating disk.

SPIF simulation is well known for the huge time required to analyze the deformation process, so many researchers have preferred to simulate only a pie model, e. g. a  $45^\circ$  pie or  $40^\circ$  pie, in the circular formed parts. In the present work, anisotropy of the material is not considered in the analyzed model and the isotropic von Mises yield criterion was assumed. The fully coupled model is simplified by applying a coupled dynamic and temperature displacement of explicit step analysis. A  $75 \times 75$  mm quarter of the  $150 \times 150$  mm sheet was taken in the simulation to decrease the number of nodes and the simulation time. In addition, the symmetry boundary condition was considered for simulating this quarter portion taken from the whole model. The mesh

elements applied in the sheet were three-dimensional solid elements of brick-shaped 8-node C3D8T thermally coupled bricks with trilinear displacement. These mesh elements are well known for their capability to simulate the heat transfer with displacement due to the high calculation accuracy. In other words, the brick element type was used because of the capability to simulate the through-thickness deformation that occurred in the SPIF method. Furthermore, the brick element is capable of deforming uniformly during the SPIF process, and SPIF is well known for the high plastic strain values. The tool and heating disc were considered as rigid non-deformable elements. From the initial analyses, the fully coupled method was selected and used as the basic simulation process in the research.

#### IV. CONCLUSIONS

Hot SPIF FEM method was performed to simulate the behavior of DP1000 sheet metal. Therefore, the hot SPIF was carried out by heating the sheet during forming with a

circular disc that moved simultaneously with the forming tool. The main simulation results can be concluded as follows:

- a. The difference between FE simulation and experiment in the Fz values during forming at room temperature was 4%.
- b. The highest stresses were found in the Z-axis and were also located a compressive stress type.
- c. The PEEQ value increased by increasing the depth of the formed part. At the same time, the PEEQ and plastic strain component were increased by increasing the heating temperature during forming.
- d. Thinning of the formed wall angle occurred in both the experiment and simulation process due to the stretching stress.
- e. Reverse bending appeared during simulation, whereas during the experiment the reverse bending was very small.