

## MODELLING AND FINITE ELEMENT SIMULATION FOR FDSC MULTI-POINT FORMING BASED ON ABAQUS

WANG WEIWEI, JIA BINBIN, SUN TINGTING, DI HE, ZHAO CHEN

School of Materials Science and Engineering  
Harbin Institute of Technology at Weihai  
264209, Weihai, Shandong, China

e-mail: wangww@hitwh.edu.cn, joybee08@126.com, stt\_hit@163.com, dihehada@163.com,  
zhaochen1994vivi@163.com

**Keywords:** Modeling, FEM, Multi-Point Forming(MPF), ABAQUS.

**Abstract.** This paper introduces the modeling and FEM simulation of FDSC Multi-Point Forming based on ABAQUS. Python is used as programming languages to secondary develop the software and to establish a simulation platform. An additional module was developed for the automatic modeling with the functions of: (a) basic parameters input such as material performance, plate size (b) rapid assembly of MPF punches and sheet automatically (c) automatic definition of the analytical step and the output of variables and (d) automatic definition of interaction conditions, loading and meshing.

The additional module developed was used to simulate the forming of different curved surfaces to verify its efficiency and accuracy. The results showed that the pre-treatment modeling for FDSC MPF can be established automatically and quickly. The simulation can be accomplished within a receivable speed in ABAQUS. Future more, experimental deformations were carried out and the simulating results are compared with that of experiment. Comparison shows a good consistency between the experimental data and simulation data.

### 1 INTRODUCTION

FDSC MPF (Force-Displacement Separated controlled Multi-Point Forming) is a new method of sheet metal forming (Figure 1). Before forming, the symmetry surface of the target surface is formed by adjust the top position of the upper punches, and then the sheet is clamped between both the upper and lower punches by filling hydraulic cylinders. When forming begins, the upper punches move down driven by the press. The uppermost punch is pushed downward first and the sheet begins to deform in this area. At the same time lower punches in corresponding position are pulled down, liquid in the cylinders is squeeze out through the relief valve to keep the clamping force. As press continues downward, the perimeter punches

move downward successively to cause more area deforming, until all punches reach to the finish line to complete the forming.

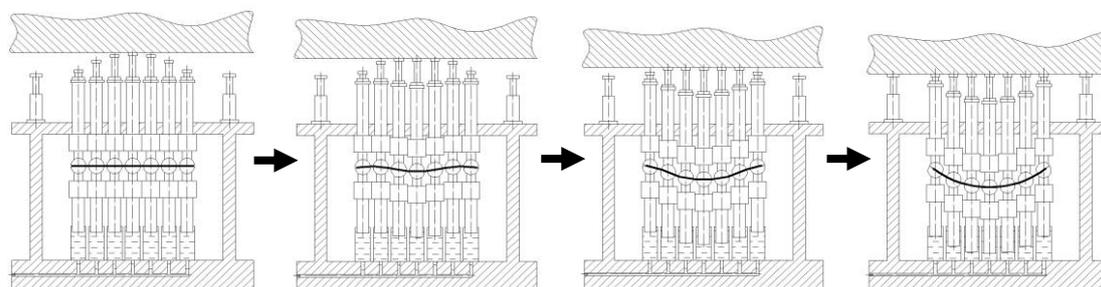


Figure 1: FDSC MPF

From the principle of FDSC MPF, it can be found that the displacement and forming force needed in the sheet metal forming are parted into two groups and controlled separately. The control method is simple and reliable which provides an adjustable and controllable normal constraint during the whole process and on the whole area of sheet metal, also greatly strengthened the constraints to sheet metal, improves the ability of forming and possesses the advantages of flexible forming.

But from the view of numerical simulation, this forming pattern brings a series of difficulties to the simulation modeling and calculation: Hundreds of punches all have independent downward movement degrees of freedom and the moving start moment and displacement varies according to the forming surface; Each head of the punches has a hemisphere hinge head which can deflect arbitrary and the direction and angle of deflecting depends on the deformation and loading state of sheet; The work face of the hemisphere hinge head keep in contact with the sheet and slid relatively in the process of deformation; Sliding also occurs between each hemisphere hinge head and the ball socket; The number, size and spacing distribution of the punches may change due to different device.....

Pei Yongsheng<sup>[1]</sup> developed a finite element parametric modeling program for traditional MPF by ANSYS/APDL with a parameter input interface, it can set up different types of finite element models through inputting parameters. Ma Hao<sup>[2]</sup> developed a parametric modeling program of FDSC MPF but the punches can not make arbitrary deflection. Quispe Rodríguez R<sup>[3]</sup> analyzed the stress distribution of linear and nonlinear analytic model's bond parts node by MATLAB and Python scripting language of ABAQUS, problem of low efficiency and low accuracy was solved. Tchalla A<sup>[4]</sup> built a multistage calculation model based on the calculation of composite material structure through writing a Python script language and user subroutine, this model can save the computer memory and CPU time.

But there is no parametric modeling program based on ABAQUS/Python which can set up finite element models of FDSC MPF with hemisphere hinge heads.

This paper introduces the modeling and numerical simulation work of FDSC MPF with hemisphere hinge heads based on ABAQUS, using Python language as a secondary development tool to develop special additional modules which used for automatic modeling and realize the automatic establishment and analysis of the model under Python environment. Comparison between the results of simulation analysis of the typical curved surface and that of experimental forming has been done to verify the accuracy and computational efficiency of the additional modules.

## 2 CONSTRUCTION METHOD OF PARAMETRIC ANALYSIS PLATFORM BASED ON ABAQUS

Parametric modeling was realized by creating plug-in programs and adds them into the main interface. Three files are needed in the programming: XX.py-the master file of modeling, XXDB.py-the graphical interface design and edit file, XX\_plugin.py-the file that link up the modeling program files and the graphical interface and create a button in the plug-ins of ABAQUS<sup>[5-6]</sup>.

The compile of plug-in program is realized by the following methods:

(1) First set up a simple model with a few punches, save the abaqus.py file and delete redundant operations, set special string as keywords in the position that scheduled to modify the parameter value, save the modified script files to obtain XX.py, all the modeling can be set up automatically on the basis of this file.

(2) One simple method to get XXDB.py is to use the RSG module in ABAQUS to add the required control and saved as a standard plug-in<sup>[7]</sup>, ABAQUS will automatically generated XXDB.py in the root directory.

(3) When creating multi-parameter dialog, link it with XX.py file, the function file XX\_plugin.py will be automatically generated in the root directory of ABAQUS.

## 3 ASSEMBLY OF THE MODEL

Function of this module: Complete the assembly modeling according to the number and arrangement of the punches. The related parameters of this module are the size and thickness of the sheet metal, the head radius, lines, columns of the punches and the spacing between adjacent punches.

Since the number, lines, columns and spacing of the punches are uncertain, the assembly patterns of the mold are of diversity. After the completion of the assembly, reference points, reference point sets and face sets needed in the modeling must be set. Every punch is independent, so how many punches have, how many reference points, reference point sets and face sets need to be established<sup>[8]</sup>.

The core program lines of the master file for Assembly module is as follows:

```
#Import the Part module of the punches
```

```

punInstance=a.Instance(name=punname,part=p, dependent=ON)
#Move the punch to the target location(x, y, z)
a.translate(instanceList=(punname,),vector=(x, y, z))
#Establish reference point in the center of the punch head
a.ReferencePoint(point=(x, y, z))
#Set contact surfaces on the punches with the sheet metal as the reference surface and named in a specified
manner
topFace = punInstance.faces.findAt(((x,y,z),))
a.Surface(side1Faces=topFace, name=facename)

```

#### 4 CONTACT CONDITIONS

The function of interaction module is to set up the contact relationship in forming process. In the numerical simulation of the MPF, the contact is mainly between the punch head and the sheet metal<sup>[9]</sup>. Each punch moves independently, so every punch needs a pair of contact separately. But the contact surface with each punch on the sheet is relatively small, if a pair of contact between every punch and the whole area of the sheet was set, there will be a large number of invalid contact area participate in the contact calculation which lead to large amount of calculation and lower efficiency (as shown in Figure 2, the yellow areas shows the contact surface on the punches, the red areas shows the contact surface on the sheet).

In order to improve the calculating efficiency, divide the blank to reduce the contact area in contact pairs, steps as follows: Divide the sheet into small sheet parts according to the numbers and arrangement of the punches, the number of the sheet parts is same to the punches, each punch contact with the sheet parts that under and surround it (Figure 3). This method not only ensures the possible contact area in the contact pairs but also reduces the effective area of contact and improves the computational efficiency.

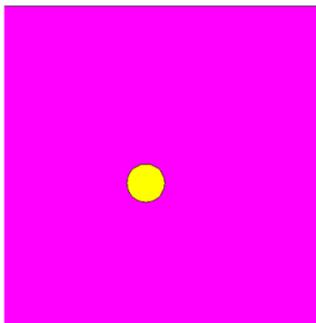


Figure 2: Contact with the whole plate

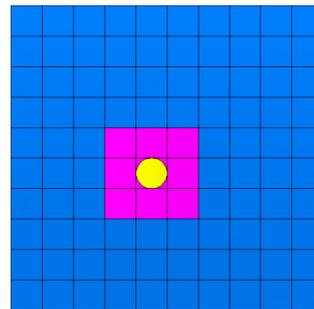


Figure 3: Contact with the divided sheet

Example of the contact between punches and sheet face:

```

#Define a contact properties and named IntProp-1
mdb.models['Model-1'].ContactProperty('IntProp-1')

```

```
#Contact types is Surface-to-surface, table=((friction,)) -contact friction coefficient
mdb.models['Model-1'].SurfaceToSurfaceContactExp(name=name3,createStepName='Initial',master=region
1,slave=region2,mechanicalConstraint=KINEMATIC,sliding=FINITE,interactionProperty='IntProp-1',initialCl
earance=OMIT,datum.Axis=None,clearanceRegion=None)
```

## 5 DEFINITION AND CONTROLLING OF PUNCHES' MOVEMENT

The function of load module is to control the punches' movement and loading. In FDSC MPF, each punch heads have to deflect along with the sheet, and different displacement curves and load curves are applied on them. The setting is as follows: The displacement is controlled by upper punches during forming, the punches only have translation along the Y direction, therefore, the displacement along X, Z direction is set to 0, the displacement adjustment along Y direction is depend on the target surface and keep the punches' X, Y, Z directions of rotational degree of freedom unset so they can rotate freely. The forming force is controlled by lower punches. The magnitude of the force in Y direction should be preset for calculation. The movement setting is same as upper punches. In this way, the lower punches can move down driven by upper punches and rotate along with sheet accompany with keep clamping on the sheet.

During forming, the upper punch moves in a uniform speed driven by the press slide. The punch that has the largest displacement moves first, the others move later. The time-displacement curves are a set of parallel straight lines with the same slope. For the punch that has the largest displacement, the time-displacement relationship is a proportion function<sup>[10]</sup>. Curve equation is:

$$y = kt \quad (1)$$

y—Displacement of the punch(mm); k—Slope of the curve; t—Movement time of the punch(s).

$$k = \frac{y_{\max}}{T} \quad (2)$$

$y_{\max}$ —The largest displacement of the punches; T—Forming time(s).

If the adjustment quantity of the punch is  $y_n$ , its displacement time curve equation is:

$$y = kt + y_n - y_{\max} \quad (3)$$

The core program lines of the master file in Load module is as follows:

```
#Define an empty two dimensional array and read the coordinates of the punches into it
yy=[]
#Set displacement-time curves according to the adjustment amount
temp001=formingtime+formingtime*yy[i][2]/bigM
mdb.models['Model-1'].SmoothStepAmplitude(name=Amplitude1,timeSpan=STEP,data=((0.0, 0.0),
(temp001, 0.0), (formingtime, 1.0)))
#Set the punches' movement mode, displacement-time curve is Amplitude1
```

```
mdb.models['Model-1'].DisplacementBC(name=flag2,createStepName='Initial',region=region,u1=SET,
u2=SET, u3=SET, ur1=UNSET, ur2=UNSET, ur3=UNSET, amplitude=UNSET, distributionType=UNIFORM,
fieldName="", localCsys=None)
#cf2=pressure, set the size of the force
mdb.models['Model-1'].loads[flag4].setValuesInStep(stepName='Step-forming',cf2=pressure)
```

## 6 MESHING

Meshing is an important part of the process in MPF parametric modeling, both the mesh unit type and grid size affect the accuracy and reliability of analysis results directly. Punches meshed with free meshing technique, grid cell is Tet. Sheet takes structured grid technology, grid cell is Quad. Variable parameters are the grid size of the punches and the sheet<sup>[11-12]</sup>.

The core program lines of the master file in Mesh module is as follows:

```
#Set up the punches' unit grid type, free meshing technique and the size of the grid cell
p.setMeshControls(regions=pickedRegions,elemShape=TET,technique=FREE)
p = mdb.models['Model-1'].parts['Pun']
p.seedPart(size=punsize, deviationFactor=0.1)
#Choose the sheet to mesh and set the grid cell size
p = mdb.models['Model-1'].parts['Sheet']
p.seedPart(size=sheetsize, deviationFactor=0.1)
```

## 7 GRAPHICAL USER INTERFACE

When the parametric programming of function modules are completed, a graphical user interface is designed by extracting the parameters related to the modeling. Values of each parameter can be input through the interface and then the related model will be established quickly by the program. The model can be modified directly by changing the parameters. The saddle surface forming is taken for example to verify the effectiveness of the analysis platform. Parameters input interface is shown in Figure 4. The forming process is shown in Figure 5 and the saddle face obtained by actual experiment is shown in Figure 6.

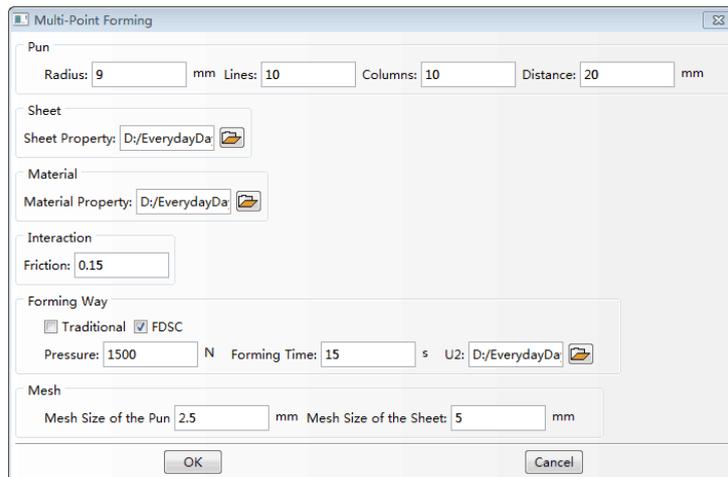


Figure 4: Graphical user interface and the parameters input

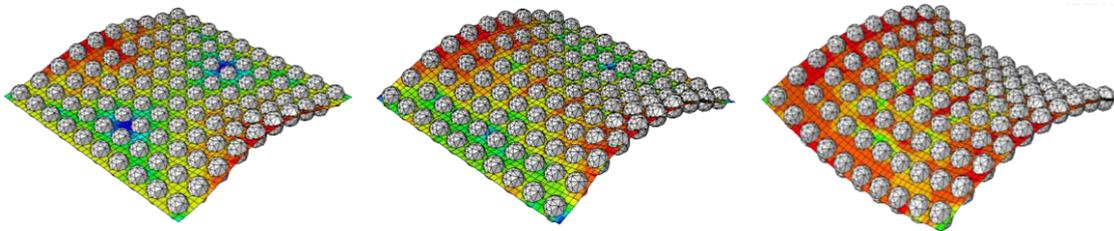


Fig5: Simulation forming process of the saddle face

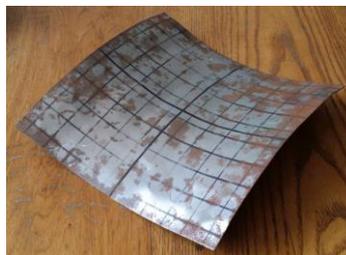


Fig6: Saddle face in actual experiment

## 8 CONCLUSIONS

A parametric analysis platform based on ABAQUS is established. Rapid assembly of punches and sheet can be realized and definition of interaction conditions, loading and meshing can also performed automatically through inputting parameters.

With this platform, both FDSC MPF model and traditional MPF model of can be established and the forming process can be simulated efficiency under the environment of ABAQUS. Modeling time is greatly saved and modeling accuracy is improved.

Results show that movements of the punches can be controlled during forming and

complex contact conditions in MPF model can be built accurately, simulation results and the experiment forming results have a good consistency.

## ACKNOWLEDGEMENTS

This work was supported by the National Natural Science Foundation of China under grant No. 51175109.

## REFERENCES

- [1] Y.S. Pei, X.P. Yan, M.Z. Li, etc. LS-DYNA Secondary development software for multi-point forming numerical simulation[J].
- [2] H. M. Design and verification of the parametric simulation program for multi-point forming based on APDL[D]. *Harbin Institute of Technology*, (2010).
- [3] Quispe Rodríguez R, de Paiva W P, Sollero P, et al. Failure criteria for adhesively bonded joints[J]. *International Journal of Adhesion and Adhesives*, (2012), 37: 26-36.
- [4] Tchalla A, Belouettar S, Makradi A, et al. An ABAQUS toolbox for multiscale finite element computation[J]. *Composites Part B: Engineering*, (2013), 52: 323-333.
- [5] J. Li, K. Yang. Study on spinning process of cylinder Based on the secondary development of Abaqus [J]. *Journal of forging technology*,( 2009), 34 (6) : 129-132.
- [6] Y. S. Jiang, Z. Y. Shi. Secondary development of ABAQUS and its application in structural analysis [J]. *Jiangsu Airlines*, (2010) (S2) : 133-135.
- [7] L. P. Sun, J. Zhou. Secondary development of ABAQUS application in design and analysis of marine flexible riser[J]. *Journal of Marine engineering*,( 2012), 33 (6) : 88-91.
- [8] Y.S. Pei, S. H. Li, Y. X. Zhao, etc. Study of multi-point forming (MPF) finite element parametric modeling [J]. *Journal of Science and Technology of He Bei University*, (2013), 33(6): 564-568.
- [9] Mustafa Yasar, Zeki Korkmaz, Muammer Gavas. Forming sheet metals by means of multipoint deep drawing method [J]. *Materials and Design*, (2007), 28: 2647-2653.
- [10] Y.S. Pei, J. G. Peng, M. Z. Li. Punches' adjustment in the process of multi-point forming technology [J]. *Journal of mechanical engineering*, (2008), 44 (1) : 150-154.
- [11] Y. F. Liang. Multi-point forming surface generation method based on triangle mesh and software development [D]. *Jilin University*,( 2009).
- [12] B. Y. Li. Research on the parameters optimization of sheet metal multi-point forming (MPF) process [D]. *Jilin University*, (2007).