Simulation Analysis for Forming and Progressive Die Design of Backing Plate

WEN Xue-hong[1], WU Yuan-ping[2], YANG Sheng-rui[3]

1. Department of Mechanical and Electrical Engineering, Foshan Polytechnic, Foshan, Guangdong 528137, China
2. Hopshing Electric Industries Co. Ltd., Foshan, Guangdong 528132, China
3. Intelligent CAD/CAM Technology Ltd., Shenzhen, Guangdong 518001, China

Abstract: The process of sheet metal stamping forming is complex, the forming process is analyzed and the blank needs to be flattened before the stamping die design. Taking a backing plate of an electric kettle as a forming process example, the part was simulated in 3DQuickForm software. The thickness, stress and strain distribution map of the forming process and blank contour is obtained. On this basis the progressive die was designed, and trial productions were carried out. The simulation results were compared and analyzed with tryouts. It verified the correctness of the simulation results. This workflow proves that simulation analysis has played an important role in improving the precision of tool design and reducing the number of physical tryouts.

Keywords: 3DQuickForm, stamping forming, simulation analysis, progressive die

1. Introduction

Metal industry tool designers rely on experience to design their tools. Not every tooling they can meet the product requirement. Consequently it requires a significant number of physical tryouts which cost a lot of manpower and material and machine costs. The overall development cycle of the tooling is also longer [1, 2]. Using finite element method numerical simulation in sheet metal stamping forming process can provide scientific basis for tool design. Simulating the processes of wrinkle, thinning, springback and cracking plays an increasingly important role to optimize the process and die structure. It achieves better tooling quality and longer service life of the whole dieset [3, 4].

Among many CAE software in the market, 3DQuickForm from 3D Quick Tools Limited provides a set of simulation tools running on SolidWorks platform. 3DQuickForm can simulate the deformation process reporting the change of the physical quantities such as thickness, stress and strain. It can quickly locate forming problems. Tooling department can develop new process scheme based on the simulated results.

2. Forming process in stamping

Fig.1 shows a backing plate of an electric kettle holder. It is a small general work piece. Considering the shape and size of the part, clearance between stamping processes, the strength of the parts, and the reliability of the processing methods, precision, efficiency, maintainability, and many other factors [5], 5 stations were set up for the part with ladder-style carriers. The basic processes include (1) Removal of peripheral material of the part, guide hole, three punch holes, & two round-extrude holes. (2) Removal of boundary material. (3) Two extrude holes. (4) Flanging. (5) Cutoff.

In the above process, extrude holes and flanging are the key processes. The deformation is affected by the tangential forces in the stretching deformation. Cracking may occur easily near the edge. With this consideration, 3DQuickForm is used to spot out unreasonable design features leading to crack occurrence as well as calculating the billet size.

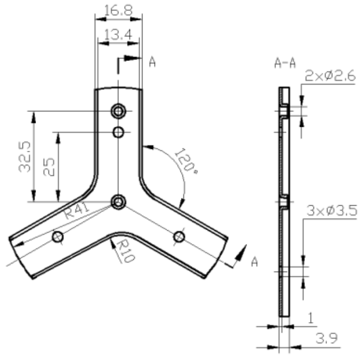


Fig. 1 Electric kettle backing plate

3. The finite element modeling and simulation

3.1 Establishment of finite element model and parameter setting

A 3D model representing the backing plate is created in SolidWorks. The neutral surface of the model is extracted and the holes are filled up. Boundary conditions like fixed edges are defined. According to the manufacturing processes and the ease of simulation, extrude holes and flanging are simulated separately. All the simulations are conducted within single SolidWorks environment without the need of data conversion.

According to the material flow rule and hardening rule, using Krupkowsky formula to describe the stress and strain behavior:

$$\sigma = K(\varepsilon_p + \varepsilon_0)^n, \quad \varepsilon_0 = \sqrt[n]{\sigma_y / K} \quad (1)$$

K --The hardening coefficient

ε_0 --The initial yield stress strain

ε_p -- The effective plastic strain

n -- Hardening index

σ_y -- The yield limit stress.

Backing plate material:

08AL with galvanized surface. Sheet metal material parameters set up in the 3DQuickForm: 1 mm thickness, Young's Modulus : 206 GPa, Anisotropic parameters : 1.65， K = 586 MPa, ε_0 = 0.007, n = 0.234.

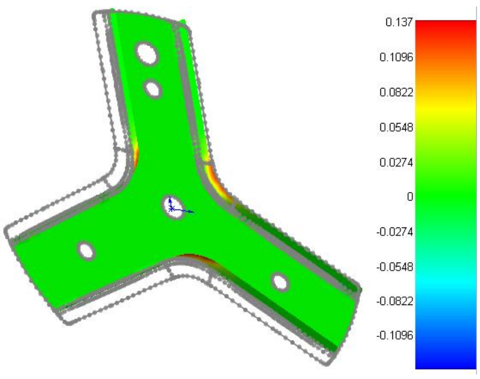
3.2 Mesh division

To mesh the finite element model, the mesh number, density distribution and quality affect simulation accuracy and computing time. The higher the mesh density, the simulation result is closer to the real value but requires more computing time. The meshing density should be increased at the important features with big deformation in order to optimize the result against computation time. [6].

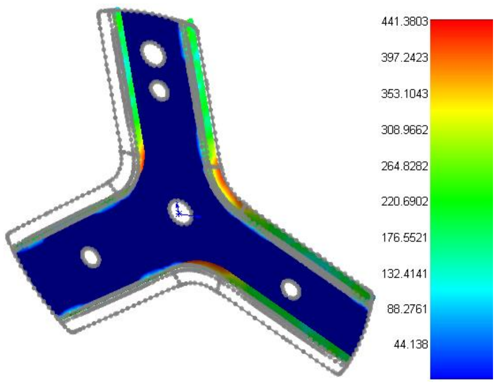
In 3DQuickForm, global mesh size is set of standard grid size to the whole part body. Setting this value bigger will shorten the overall computing time. Minimum mesh size is set for the detailed features such as surface angle, bending radius, and forming angle. Taking into account the forming process of the sheet metal parts, the mesh size is adjusted with respect to the thickness of the sheet metal parts. For the initial run, the mesh size is generally set to be 1/10~1/20. This quickly determines whether the part has error in meshing or cannot converge into result. After the first analysis, the mesh is refined. For the second run, the mesh size is the half of the initial mesh size. The global mesh size is 0.05 mm, and the minimum mesh size is 0.02 mm.

3.3 Flanging simulation results and analysis

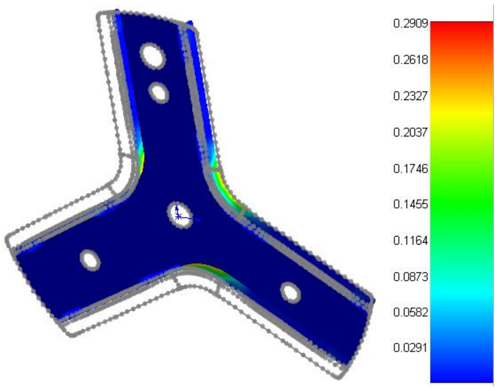
Flanging forming simulation results are shown in fig.2. From the simulation results at the edge of the flange, maximum material thinning at 0.137 mm, maximum stress at 441.38 MPa, the maximum strain at 0.29. All within the material allowable tensile limits and no cracking will not take place. Flanging quality is very good and meets the specification. At the same time, the analysis finalizes the developable contour of the billet.



(a) Thickness distribution



(b)The stress distribution

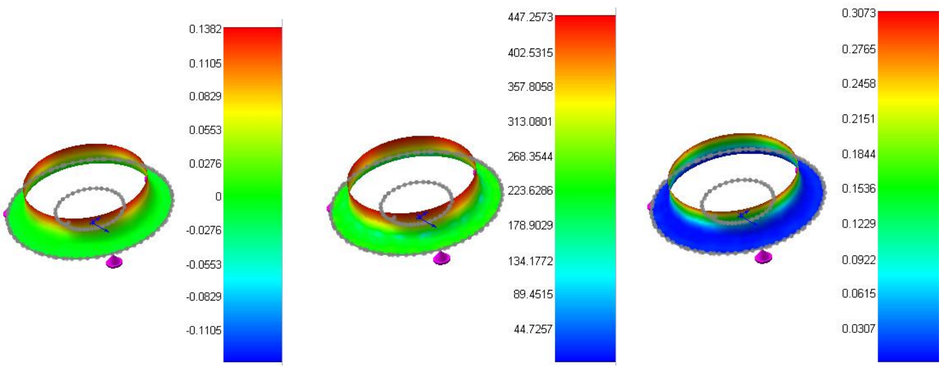


(c) The strain distribution

Fig. 2 Results of flanging forming simulation

3.4 Simulation results and analysis of the extrude holes

The simulation results of the extrude hole forming are shown in fig 3. From the simulation results, deformation occurs at the edge of the hole with maximum thinning of 0.138 mm, the maximum stress of 447.25 MPa, and the maximum strain of 0.30. All are within the material allowable ultimate tensile range, no crack will take place. Hole flanging is good quality meeting the specification.



(a) Thinning distribution (b) The stress distribution (c) The strain distribution

Fig. 3 Simulation results of the turning hole forming

4. Tool design

According to the results of 3DQuickForm analysis, the forming process parameters are determined, and the layout of the blank size is calculated using 3DQuickForm in SolidWorks, as shown in Fig. 4.

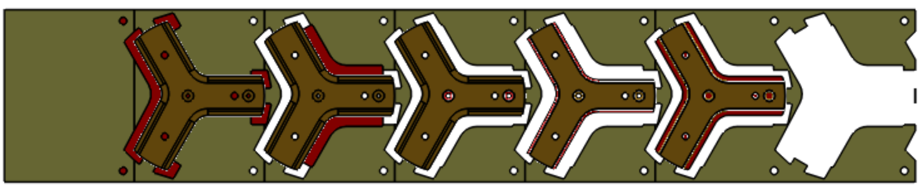


Fig.5 Layout design

In order to guarantee precision tooling, guide pins are applied. The strip position is locked to keep the precision requirement. 7 set of floating guide pin and three lifters are used to facilitate the strip material passing and part cutoff. The cutoff performance is achieved by employing better performance spring stripper for stripping the part as shown in fig 5.

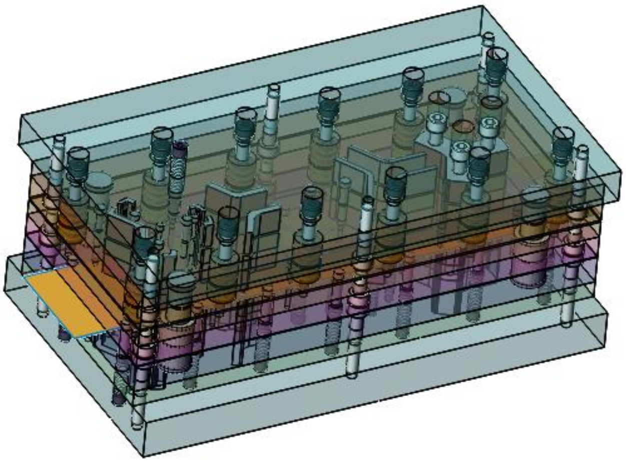


Fig.5 Three-dimensional structure of the tooling

The punches and dies need to have good wear resistance and strength. Together with the striper plate, Cr12 cold forging steel is used. C45 steel is used for the remaining components. The initial trial out is shown in fig. 6. Tryout results resembled to the simulated result at high degree.

Fig.6 Tryout part

5. Conclusion

(1) The forming simulation analysis of the stamping forming process of the backing plate of the electric kettle is carried out by 3DQuickForm. The change of the thickness, stress and strain of the part is reflected by the visual images, which provides very good reference for the die design.

(2) 3DQuickForm calculates billet size, optimize the blank layout to get a reasonable blank shape.

(3) According to the forming simulation to decide the stamping process parameters, it helps the full process of design and manufacturing in progressive die manufacturing. Backing plate is produced according to the specification.

In conclusion, tool design based on finite element analysis gets a reasonable stamping process and parameters. It avoids the repeated physical tryouts, minimizes tooling modification, shortens the tool development cycle, and improves the quality of the parts.

Author Background：
Ms. WEN Xue-hong（1975-）
Jinxian City Jiangxi Province, PRC
Lecturer and Master in Mold & Die teaching and research works