Computer Life (CPL) ISSN: 1819-4818
DELIVERING QUALITY SCIENCE TO THE WORLD

Numerical Simulation of Drawing Forming of Back Beam of Top Cover

Mingming Zhong [a], Longyu Ma [b, *]

Shandong University of Science and Technology, Qingdao, 26000, China.

[a]2675415682@qq.com,*, [b]1726605521@qq.com

Abstract: This paper discusses the development and current situation of mould industry in China, introduces the application of UG in mould design, and discusses the whole process of mould design of the beam after cover in detail. According to the shape and technology requirement of the parts, the process analysis was carried out, and the stamping forming process scheme of the parts was formulated. This design using Dynaform software to carry on the analysis, through to the drawing, the analysis of the springback, wrinkling, etc, check the mould structure design is reasonable, timely improve mold structure, ensure the quality of workpiece. With the help of computer simulation analysis, the time required for mold design is greatly reduced, the design cost is reduced, and the production efficiency is improved. Paper introduces in detail after the roof beams of the whole process of mould structure design and key parts design (including blanking force, discharge power and calculation of the output of the spring, the choice of the unloading screws, convex and concave die design, the upper and lower mold base, the choice of the guide pin, guide sleeve, etc.). The selection of stamping equipment was briefly introduced, and the manufacturing process of main parts of the die was worked out. The process of mould design and 3d modeling using UG are briefly described.
Keywords: drawing, trimming punch, flanging punch die design; Dynaform finite element analysis; Process analysis; the structure design.

---

1. INTRODUCTION

Metal plastic forming process is affected by many factors, such as material performance, mold shape, blank shape, process parameters. In the past, we mainly relied on experiments, experience or theoretical analysis under the premise of more simplification and hypothesis. However, such theoretical analysis results and actual results are often very different, so in the practical application of large restrictions. Because reducing the design cycle, mould cost and stop-time is an effective method to reduce the cost of workpiece, and these are based on the familiar material characteristics in the forming process, so the designer has a higher requirement. In order to achieve these goals accurately and quickly, numerical simulation becomes very important [1].

With the increasing popularity of computing technology and computer application, especially the continuous improvement of finite element technology, the numerical simulation technology of metal plastic forming process has been rapidly developed, which makes the theory of plastic forming step forward to practical application. For the specific products and materials, the numerical simulation technology can be used to quantitatively analyze the stress and strain distribution of various forming processes, understand the law of plastic flow, the distribution of temperature field and the effect of various deformation parameters. It is because of the wide adaptability of the finite element analysis method that it becomes an indispensable part in the study of plastic forming theory.

In this paper, the finite element analysis of the drawing forming process of the rear peripheral plate of the van is carried out by using the software DYNAFORM.

2. PROPERTIES

The application of finite element analysis to plastic machining is just developed in the last 20 years. As far as metal plastic forming is concerned, the finite element method can be roughly divided into two categories:

One is Solid plastic finite element method, consisting of small deformation and large deformation elastoplastic finite element method. Elastoplastic finite element was first proposed by Marcal and King in 1967[2]. It considers both elastic deformation and plastic deformation, and adopts Hook law in the elastic region, plandl-reuss equation and Mises yield criterion in the plastic region. Based on the small deformation theory, the small deformation elastoplastic finite element method ignores the local deformation of the microelement and considers that the displacement and strain show a linear relationship and is only suitable for analyzing the initial stage of metal plastic forming. Based on the finite deformation theory, the elastoplastic finite element method takes into account the influence of large displacement and large rotation on the element shape and finite element calculation in the process of large deformation. Using elastic-plastic finite element method (fem) analysis of metal plastic forming process, not only can get change of plastic zone according to the deformation path, deformation of stress and strain distribution rule and the change of the size and geometry, but also can effectively handle unloading problem, calculate the residual stress and residual strain, and then compute and defect prediction of springback analysis. Elastic-plastic finite element method (fem), however, because want to consider the correlation of deformation through Sue, the incremental load, should be used in in each incremental step loading, are required to make the elastic calculation to determine whether the original unit in the elastic area has entered the yield, yield to enter after the unit is using elastic-plastic constitutive relation, thus changing the element stiffness matrix. In order to ensure the convergent parts of precision and reconciliation, each load cannot make many units yield at the same time, which makes the deformation increment during each calculation cannot be too large, so the calculation time of large deformation problem is long and the efficiency is low.

The other type is Flowing plasticity finite element method, including rigid plasticity finite element method and rigid viscoplastic finite element method. In this kind of finite element method, the Levy-Mises equation is adopted as the constitutive equation, which satisfies the volume invariant condition [3]. By using the rate equation, the shape of the deformed object is obtained by integrating the velocity in the discrete space, thus avoiding the geometrical nonlinearity problem in the finite deformation. At the same time, a larger incremental step length than the elastoplastic finite element method can be used to reduce calculation time and improve calculation efficiency, and sufficient engineering precision can be guaranteed. However, due to the neglect of elastic deformation, this kind of finite element method cannot deal with unloading problems and calculate residual stress, residual strain and resilience.

3. FINITE ELEMENT SIMULATION OF DRAWING FORMING OF THE BACK BEAM OF TOP COVER

3.1 Analog process analysis

There is no difficult part during forming, the shape is simple and smooth, and the negative Angle of stamping will not appear by selecting a reasonable direction of stamping. Reasonable punching direction should ensure that the punch can complete the part that needs to be drawn deep in the first drawing, and there should be no dead Angle or dead zone that the punch cannot reach. At the beginning of drawing, the contact state between the punch and the blank is good[4], the contact area should be as large as possible, the contact point should be close to the middle of the shape, and the contact point of the blank should be more and dispersed at the same time. And draw the depth of uniform, uniform feed resistance. The workpiece can be made by a single drawing forming method. In order to make up for the deficiency of the workpiece in stamping process, the workpiece body part should be added before drawing forming. There are two types of technical supplementary faces here: one is the internal technological supplement of the part that is, filling the internal holes. The technical supplementary faces can be removed through the punching process after drawing forming; The second part is the external process supplementary surface of the part along the edge of contour line [5]. The pressing surface should be conducive to reducing the drawing depth. A certain geometric relationship should be maintained with the shape of the punch to ensure that the blank is in a state of tension during the drawing process, and can smoothly and gradually adhere to the punch to prevent wrinkles. In addition, the pressing surface should be a plane or a smooth surface, and there should be no local drum, pit, folded edge, and no creasing when the blank is pressed. The design of drawbead should arrange one or more drawbead to increase the resistance of feed and improve the plastic deformation degree of the material. A local short rib is provided at the location where it is easy to wrinkle to increase the radial tensile stress, reduce the compressive stress and prevent the preform from wrinkling [6].

3.2 Mesh division of curved surface

(1)The reselection menu Preprocess to Element, and click the "Surface Mesh" button (the fourth column in the first row) in the pop-up Element dialog box.

(2)In the "Surface Mesh" dialog box, the maximum unit value (max-size) is set to 20, and the values of other items are automatically divided into grids by default. The grid cells are shown in figure 3.1.

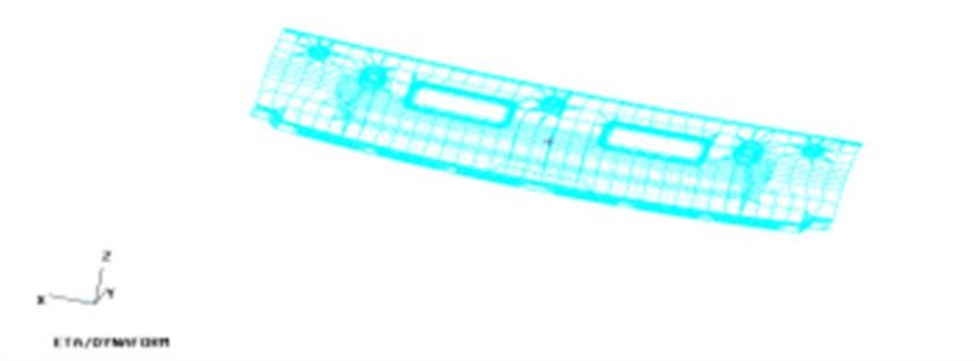

Fig 3.1 grid cells

4. CONCLUSION

(1) The final forming results are shown in figure 4.1.

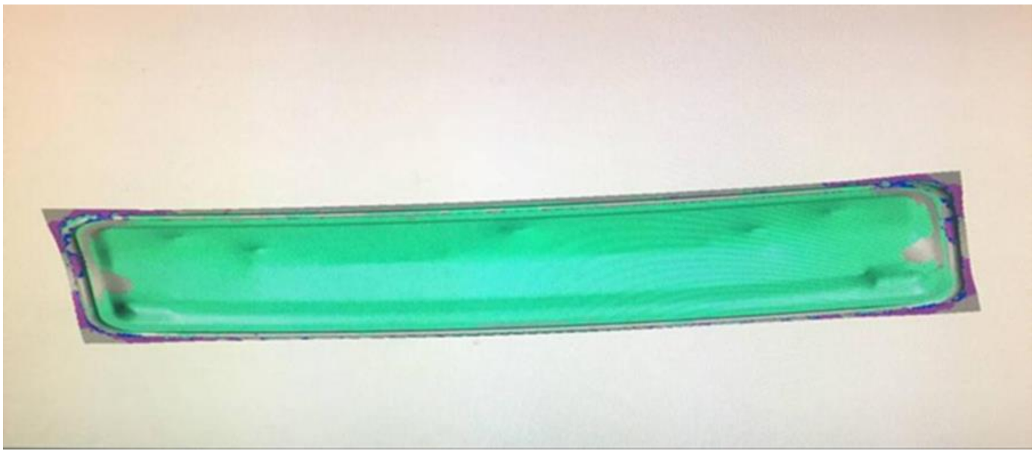

Fig. 4.1 formation results

(2) Parts forming limit diagram is shown in Fig. 4.2.

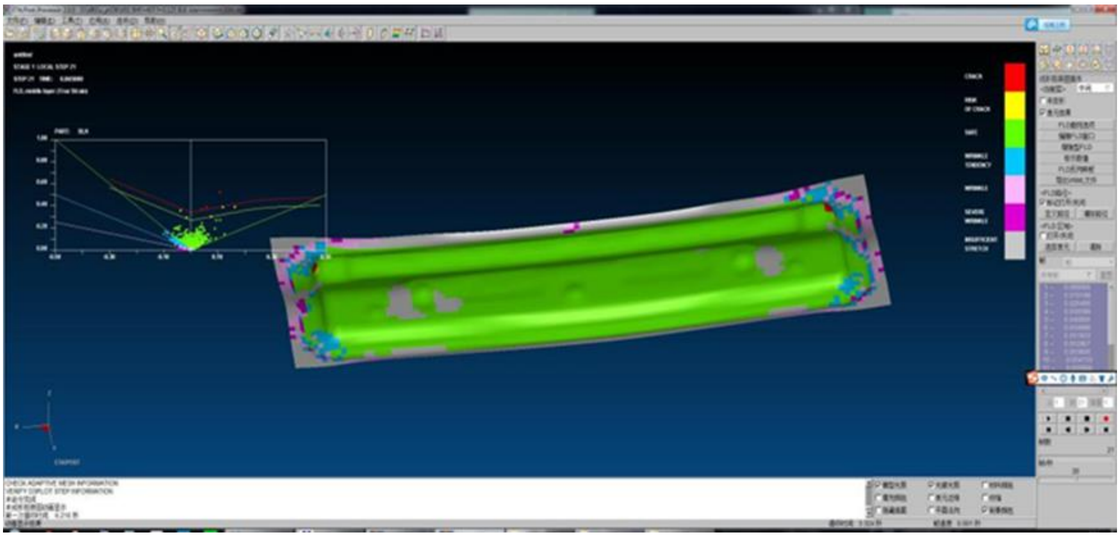

Fig. 4.2 forming limit diagram

According to the simulation results, FLD (forming limit diagram) was studied, and it was found that there was wrinkle state on the edge, which did not have a great impact on product quality and use in actual production. The key part belonged to the safe forming area, and there was no wrinkling. Therefore, this product could be formed in the study of forming limit diagram.

(3) The final thickness change of part forming is shown in FIG. 4.3.

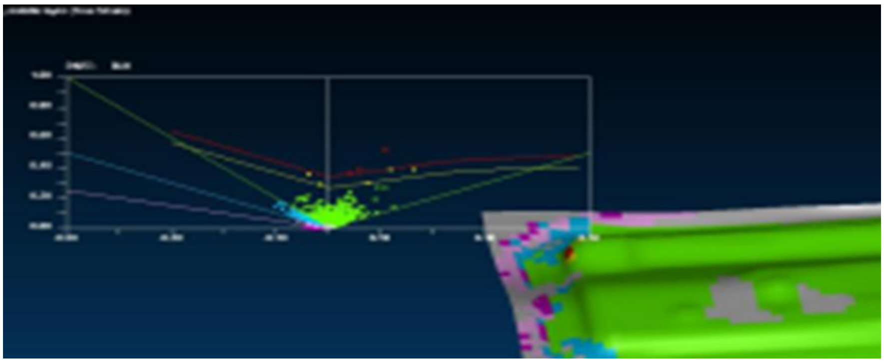

Fig. 4.3 final thickness variation

According to the simulation results, it was found that the thickness of the parts did not change very much, the thinnest part was 0.97mm, and the thinning amount was less than 30%. It could be considered that no fracture would occur. The key part belonged to the safe molding area, and the final part was 1.28mm, without serious wrinkle. Therefore, the product can be formed under the study of thickness variation chart.

(4) The curve of the pressure -- time of the protruding die is shown in FIG. 4.20.

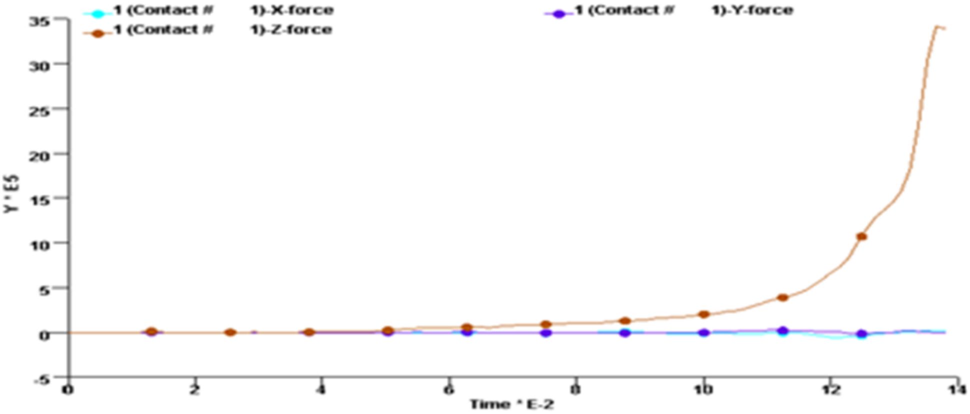

Fig. 4 13 punch pressure - displacement curve

The maximum z-axis pressure of the punch is 35 x E5 N, that is 350 tons, which can be used as a basis for selection of the press.

REFERENCES

[1] Amano, T., Tamura, K., 1984. The study of an elliptical cone spinning by the trial equipment. In: Kobayashi, M. (Ed.), Proceedings of the Third International Con-ference on Rotary Metalworking Processes. Kyoto, Japan, pp. 213–224.

[2] Arai, H., 2005. Robotic metal spinning-Forming asymmetric products using force control. In: Proceedings of 2005 IEEE International Conference on Robotics and Automation, Barcelona, Spain, pp. 2702–2707.

[3] Awiszus, B., Meyer, F., 2005. Metal spinning of non-circular hollow parts. In: Bariani, P.F. (Ed.), Proceedings of the Eighth International Conference on Technology of Plasticity. Verona, Italy, pp. 353–354.

[4] Gao, X.-C., Kang, D.-C., Meng, X.-F., Wu, H.-J., 1999. Experimental research on a new technology-ellipse spinning. J. Mater. Process. Technol. 94, 197–200.

[5] Wong, C.C., Dean, T.A., Lin, J., 2003. A review of spinning, shear forming and flow forming processes. Int. J. Mach. Tools Manuf. 43, 1419–1435.

[6] Zhan, M., Yang, H., Zhang, J.H., Xu, Y.L., Ma, F., 2007. 3D FEM analysis of influence of roller feed rate on forming force and quality of cone spinning. J. Mater. Process. Technol. 187–188, 486–491.