Aid Design Die of Auto-Body Using Numerical Simulation of 3-D Sheet Metal Forming Processes

Ninig-an Hu
SAE China (Society of Automotive Engineering of China)
YUEJIN AUTOMOBILE GROUP CORPORATION TOOL PLANT
HEIMOYING ZHONGYANGMENWAI NANJING CHINA
Tel: 086-025-5417698
E-mail: huningan@jlonline.com

Ning-yan Zhu
SAE China (Society of Automotive Engineering of China)
PLANNING AND DEVELOPMENT DEPARTMENT OF NANJING IVECO LTD
(SINO-ITALIAN JOINT-VENTURE)
100, HEIMOYING XUANWU DIST NANJING CHINA
Tel: 086-025-5417711-54998
E-mail: huningan@jlonline.com

Abbreviations: FLD: form limit diagram

Keywords: DIE, DRAW-BEAD, MESHING

ABSTRACT

It has been well applied in automotive manufacture of China about the technology of the numerical simulation of 3-D sheet metal forming processes during the recent years.

Besides helping tool designers and artificers find optimal shapes of technical surface on the die, DYNAFORM is used in the early stages of the auto-body to work over the formability of various parts. In addition to the stamping process, the new forming technologies can be analyzed with DYNAFORM to find more efficient methods and optimal processing parameters for parts of the auto-body.

The paper discussed how DYNAFORM is used to select a critical part in my plant and demonstrated how numerical simulation can reduce the developing time and costs of tools.

For example, the formability analysis for the drawing die of the inside rear door on the car was performed using the numerical simulation of 3-D sheet metal forming processes. It is well known that the occurrences of both fracture and wrinkle were the two encountered difficulties in the stamping process. The numerical simulation was performed to analyze the metal-flow that caused the fracture and wrinkle on the draw-bead and it also acted to analyze the change of the blank's outline that could bring up the fracture or wrinkle. The strain distributions were obtained from the numerical simulations and were also used in conjunction with the forming limit diagram to predict the onset of the fracture. The effect of blank-holder pressure and friction on the occurrence of the fracture and wrinkle was researched. To prevent the formation of the fracture and wrinkle, according to the above formability analysis of the numerical simulation, an optimum shape of the technical surface, the draw-bead distribution and sizes on the drawing die surface and the blank's outline were finally determined.

INTRODUCTION

A successful die design generally results from an accurate prediction of the metal-flow during the forming process, which relies mostly on the experience and know-how of the designer in the actual practice. However, the FEM was applied to the formability analysis of the 3-D sheet metal forming processes, the metal-flow could be easily predicted from the numerical simulation. First, the sheet metal forming analysis of the stamping process using DYNAFORM is to avoid the very costly and consuming time in the design phase by predicting key outcomes such as the final shape of the technical surface, the possibility of fracturing and wrinkling, and metal-flow. Such 3-D numerical simulation is most useful and efficient when it is performed in the early stage of design by designers, rather than by analysis specialists after the detailed design is complete. Second, it reduces manufacturing iteration, testing and improves the punches and dies. The sheet metal forming is a process that is mainly characterized by contact. Inherent in such processes are rapid changes in contact regions, contact of deformable parts with rigid tools and large relative motions of material on contacting surfaces. The sheet metal forming also involves large stretching and bending stresses that may require appropriate constitutive material models. The sheet metal forming analysis of the stamping process is not straightforward to find a set of feasible solutions that satisfies these goals. A great deal of time and money in parts of automotive bodywork is thus consumed by finding appropriate tool geometry and manufacturing parameters by trial and error, whereby physical experiments must be performed and tools are repeatedly modified according to the experimental results.

The aim of the paper is to demonstrate that DYNAFORM can be used successfully to simulate the sheet metal forming process for a wide range of configurations commonly encountered in real forming parts of automotive bodywork.

APPROACH

Although the sheet metal forming processes are diverse, the primary objective is to produce a desired changing shape. These operations frequently involve a complicated three-dimensional deformation, which means a combination of stretching, drawing toward the complex surface (e.g. in the inside rear door on the car of stamping problem as shown in Figure. 1). Consequently, the properties of the sheet metal are a paramount importance and require modeling by means of a very precise constitutive sheet metal law. In order to

Figure 1. Actual inside rear door on the car

facilitate the analysis, a series of experiments were conducted to obtain the sheet metal properties and the forming limit diagram for the actual parts.

Pre-And Post-Processing

Most of the hands-on time for numerical simulation cycle is currently spent for parameters and shape of surface preparation, meshing meshes, and evaluation of numerical simulation results.

3-D CAD Geometric Surface. For tool surface meshing, a geometric domain of the part including the blank-holder surface should be directly imported in the IGES file format from 3-D CAD software (UG II) without the user's intervention to realize a close integration of design and analysis.

The geometry of the drawing die of the inside rear door on the car, as initially designed, was generated by commercial 3-D CAD software. The 3-D CAD file must be clean: i.e., an

Figure 2. Geometry of inside rear door on the car

overlap or gap is not allowed to exist between surfaces, so that the finite element meshes can be constructed. As shown in Figure 2.

Element Size Control. It is necessary to control the sizes of mesh elements over the entire domain. In meshing a sheet of geometry, we must consider three criteria to determine the element size distribution: (1) curvature, (2) contact between the sheet metal and tool surfaces, and (3) gradient of strain distribution. A finer mesh must be used in regions of high curvature, tool contact, and drastic change of strain distribution.

Element Shape Regularity. A sheet must be meshed into a set of well-shaped elements, because of excessively skinny or distorted elements will slow the convergence and may cause analysis errors. This requirement does not necessarily apply to tool surface meshing. As shown in Figure 3.

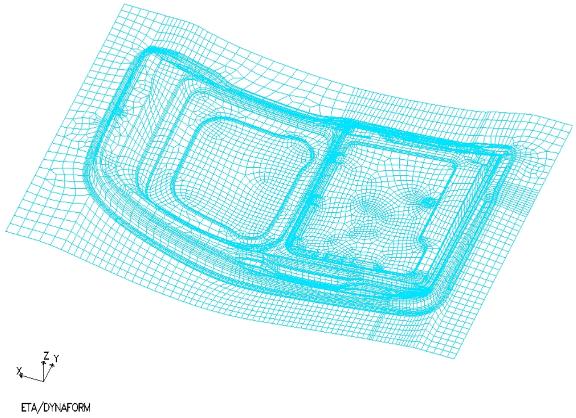

Figure 3. Meshes of inside rear door on the car

Adaptive Re-meshing. In a large deformation problem, the initial meshes may become severely distorted as the numerical simulation proceeds, yielding an incorrect solution. In such a case, the sheet must be re-meshed during the numerical simulation. Adaptive re-meshing is also necessary when critical portions of sheet metal forming analysis, which require finer mesh element, change drastically over time as the numerical simulation proceeds. In this case, adaptively refining the critical areas and coarsening non-critical areas of the mesh help to reduce the computational time and improve solution accuracy.

Parallelism. Like FEM simulation itself, the meshing process should be broken down into smaller, parallel meshing tasks. A simple way to do this is to decompose the whole domain into multiple sub-domains, each of which is then meshed by a single CPU. However in doing this, some current meshing algorithms tend to introduce ill-shaped elements along sub-domain boundaries, possibly degrading the quality of numerical simulation. A good meshing should create well-shaped elements even when the input geometric domain is subdivided prior to meshing.

The difference in meshing requirements for a sheet of geometry and a tool surface can be summarized as follows. The challenge in meshing a sheet of geometry is to control the element sizes precisely while keeping the element shape as regular as possible; this is not an easy task, especially in quadrilateral meshing. In meshing a tool surface, on the other hand, the difficulty is in converting various types of CAD surfaces consistently into anisotropic quadrilateral with their sizes and directionality controlled on the basis of local curvature.

Evaluation. Evaluation of the simulation results is of significant importance. All relevant quantities can be graphically represented and documented. Several quantities can be traced and monitored during the process and shown as 2-D history-plots. Using FLD to detect the area of the fracture, major and minor strains for an arbitrary area of the part can be depicted in the FLD. The distance to the FLD is available as a post-value. An animation that can be generated with DYNAFORM is at times very helpful to gain more insight into the forming process.

In the numerical simulation, the sheet blank was placed on the die then the blank-holder is moved down to clamp the sheet-blank against the blank-holder, so that the shape of binder-wrap was formed. After which, the punch was continued down to draw the sheet-blank to the desired shape.

Drawing Die Design

The stamping of the inside rear door on the car is a drawing process with a complex surface because of the inherent geometry of the part. The complete set of tools (blank-holder, punch and die) can be generated from the die with offset and trimming operations in 3-D CAD software. Next, the geometry of the part including the blank-holder surface is generally given in the IGES format file. For processing in DYNAFORM, this performed nearly automatically but it is possible to modify the mesh interactively to overcome problems that may arise.

Design Technical Surface. The shape of die cavity conforming to the geometry of the inside rear door on the car also remained the same for all die surface designs since the inside rear door is drawn to the desired shape by one operation. In consequence, the drawing die design was focused only on the drawing die technical surface. In order to analyze the metal-flow, a flat surface without the draw-bead was used as the initial design for the drawing die surface. The analysis of the modified drawing die designs was then performed according to the phenomenon of metal-flow obtained from the numerical simulations.

Design Sheet Blank's Outline. An optimum sheet-blank shape was determined by the analysis of the numerical simulation was used for the drawing die design. The sheet blank's outline was also determined by using function of DYNAFORM's OUTLINE.

Design Process Parameters. To design the die, the numerical simulation was also conducted to study the effect of the process parameters. The simulation results are summarized and consulted for designs. The clamping force exerted by the blank-holder was 6000N and a coefficient of friction of 0.122 was assumed for the initial die of the numerical simulation.

Design Draw-bead. The function of the draw-bead is to control metal-flow from blank-holder to the die cavity. In order to eliminate the wrinkles without causing the problem of fracture, an optimum design of the size and the location of the draw-beads on the die surface proves itself an efficient method to achieve this goal. Following the observation of the metal-flow obtained from the numerical simulation results, so that a more precise control of metal-flow could be achieved. In the numerical simulation, the equivalent draw-bead model was used and the restraining force produced by the actual draw-bead was assumed, instead of considering the actual geometry of the draw-bead. The advantage of using the equivalent draw-bead model is two-point: one is the large saving of computation time and the other is that we do not have to deal with the actual draw-bead shapes until the optimum restraining forces are determined. The design shape of the draw-bead can then be derived from the optimum restraining force so that the draw-bead will produce. The sheet metal passing through the equivalent draw-bead model is subjected to the same restraining force as that exerted by the actual draw-bead. The use of an equivalent draw-bead model eliminates the need for using an extremely fine mesh. Therefore, an accurate estimation of the restraining force produced by the actual draw-bead is essential for setting up the equivalent draw-bead model. The draw-bead lengths and the restraining forces produced by the draw-bead have been repeatedly modified according to the analysis of metal-flow obtained from the numerical simulations until an optimum combination is determined.

DISCUSSIONS OF RESULTS

In the actual stamping operation, there are several possible steps to obtain the metal-flow described, such as increasing blank-holder pressure, increasing friction at the interface between sheet-blank and die, and adding draw-beads on the die surface. In addition to the die surface design, the numerical simulation was also conducted to study the effect of the process parameters, such as friction and blank-holder pressure, on the formability of the stamping process.

CONCLUSIONS

In the search for more economical manufacturing technology, forming processes will become increasingly important in the future. The numerical simulation techniques are making a significant contribution to the development of new forming technology.

Weight reduction of structures must nevertheless be realized. Therefore, the reliability of the simulation results must be increased, as design limitations become more exacting. It will thus become necessary to observe material behavior more accurately.

The use of the simulation demonstrated that close cooperation and communication between all departments concerned with the development of a part was the only way to carry out the necessary changes to the forming process and the geometry of the tools and then significantly reduce development time.

The drawing die surface design for stamping the inside rear door on the car was investigated using the 3-D numerical simulations. In the investigation, the cause of the formation of wrinkles was studied on the basis of the metal-flow obtained from the numerical simulation results. The FLD was also used in conjunction with the numerical simulation results to predict the occurrence of fracture. In addition, an equivalent draw-bead model was applied to the numerical simulations to save the computation time.

An optimum draw-bead design was then determined from the numerical simulation analysis and could be used for the actual die design.

In the present study, the effects of the process parameters on the formation of fracture and wrinkle were investigated. It is found in the investigation that wrinkles, although can be eliminated by increasing the blank-holder pressure or the friction at the interface between the die and the sheet-blank, the occurrence of fracture may be accompanied with this procedure. Due to the effect of increasing the blank-holder pressure and friction is uniformly distributed on the sheet-blank and can limit metal-flow, an optimum design of the use of equivalent draw-beads is necessary. The shapes are not considered until the optimum design is achieved, resulting in an efficient approach for the draw-bead design.

ACKNOWLEDGEMENTS

The author would like to thank Zhong-qin Lin and Wei-gang Zhang SHANGHAI JIAO TONG UNIVERSITY, Rui-quan Guo SHANGHAI BAOSTEEL GROUP CO. and He Fang YUEJIN AUTOMOBILE GROUP for providing help.

REFERENCES

KAWKA, M. and MAKINOUCHI, A. Sheet Element Formulation in the Static Explicit FEM Code for Simulation of Sheet Metal, Proc. of Numisheet' 93, pp. 97-107,1993.

SHIMADA, K., Physically-Based Mesh Generation: Automated Triangulation of Surfaces and Volumes via Bubble Packing, Ph. D. Thesis, Massachusetts Institute of Technology, Gambridge, MA, 1993.

HAUG, E. and DI PASQUALE, E., 1991, Industrial sheet metal forming simulation using explicit finite element methods, Proc. Int. VDI berichte NR. Vol.894, pp.441-466.

H. PARISCH, 1995, A Continuum-Based Shell Theory for Nonlinear Applications, Int. J. Numerical Methods in Engineering, Vol.38, pp. 1855-1883

T. BELYTSCHKO and ITAI LEVIATHON, 1994, Physical Stabilization of the 4-node Sell Element with One-point Quadrature, Computer Methods in Applied Mechanics and Engineering, Vol.113,pp.321-350.

YOU-MIN HUANG and DAW-KWEI LEU, 1995, Finite Element Analysis of Contact Problems for Sheet Metal Bending Process, Computers and Structures,Vol.57,pp.15-27.