International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 12, December-2014

Research on the Formability Analysis for Automobile Reinforcement Plate for Front Side Edge Beam Box

Emmire Wosenyeleh Sebsibe, Yu Jun Cai, Lie Duan
School of mechanical engineering,
Tianjin University of technology and education,
Tianjin, 300222, P. R. CHINA

Abstract: **Sheet metal stamping processes play an important role in automobile industries manufacturing. Reinforcements is used tocarry load of the passengers, support the load of the engine, the forces caused due to the sudden braking or acceleration, centrifugal force while cornering all need reinforcements considering with weight reduction. Finite element method (FEM) is the most common method used for this purpose. To reducing time, manufacturing costs, improving safety….etc. It is also seen that a material selection plays an important role in the formability of the part. In this study, FEA simulation is conducted to analyze and chosen appropriate process flow for reinforcement'splate for front side beam box.**

Keywords: Finite element analysis, simulation, sheet metal forming, DYNAFORM

INTRODUCTION

When we think of passenger safety systems, the one thing that usually comes to mind is the vehicles Reinforcement. Reinforcement plate for side edgebeam box is connecting of the body side edgeand the front floorof body.This component is what protects your ride's front ends from damage in the event of minor or light collisions. During a low-speed collision, for example, the reinforcement absorbs as muchforce as possible, to prevent damage on your engine bay or trunk.

Sheet metal forming has a very high importance in manufacturing in automobile industries. In the last decade, due to the inventions and developments of the new materials and their high quality and enhanced functionality, sheet metal forming became more popular, its application reas are extremely increased and it turned out to be one of the most preferable manufacturing techniques in industry around the world [1]. Forming characteristics such as thinning, rupture (splitting), wrinkling, etc., can be predicted and controlled with high level of confidence using the CAE tools such as Abacus/, Hyper Form (Altair Hyper works), DYANFORM/LS-DYNA (ETA/LSTC) etc.

With the help of FEM simulation, amendment of the preliminary or final design of the Product die after the tool try-out can be avoided. Others forming parameters, which would result in a product with the required quality, can be determined in an efficient way.

Process methodology

To predictthe analysis and results values flow of methodology is shown as followed in Fig. (1)

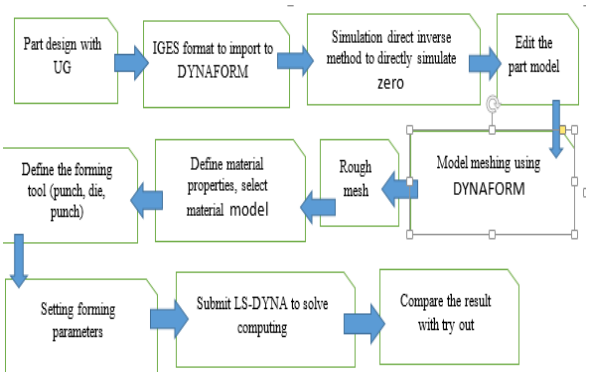

Fig. (1) Process flow of the methodology

Simulation modeling

In this study of reinforcement plate for front side beam box as shown in Fig.(2)of this final shape CAE software (UG NX8) is used to generate the model geometry and import IGES data to transfer to CAE(DYANFORM) software. The blank material is Material -Type36 in DAYNAFORM: QC mild steel which properties are the same with the real object .The blank thickness is 1.6mm. The dimensions of this part are 210 mm by 136mm with part depth of 50.727mm. DYANFORM is the complete die system simulation software. Utilization of it allows the

organization to entirely bypass soft tooling, reducing overall tryout time, lowering costs, increasing productivity and providing complete confidence in die system design [2]. In this part we offset tool surface meshing, all geometric part (die, punch and binder) from die face Engineering (DFE) module and estimate blank with Blank Size Engineering (BSE) module.As can be seen from Fig.(2) edge beam floor around the car side box reinforcing plate is a symmetrical parts, parts structure is relatively simple, some curvature changed greatly, but most of the surface is flat. Parts After drawing, punching, trimming, flanging, shaping and other stamping processes. Drawing is a key process in which the paper is only part of its drawing .Drawing process part model shown in Fig. (3).Forming process is of the analysis, the subsequent trimming, flanging, shaping and other processes come to a complete qualified parts. Based on the product design of a reinforcement, the DFE module offers capabilities of both CAD surface and CAE meshing tools. DFE interactively generates binder surfaces, addendum profiles/surfaces, PO lines and layout draw beads with full associativity between FEA mesh and surfaces. A preliminary die face is created for further formability studies with an iterative process until die face validation is achieved [5]. tomeetthe requirements ofthe drawing process, in order to achievethe drawing, to make up fordeficienciesinthe partformingprocess, so that partis easyshapingoftenadditionallyaddaddendumoutside ofthe bodyparts .Afteraddingaddendum the partgeometryshown in Fig.(4).

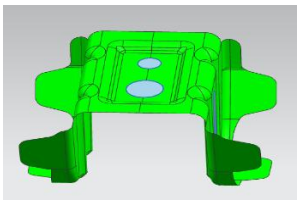

Fig. (2)UG main model

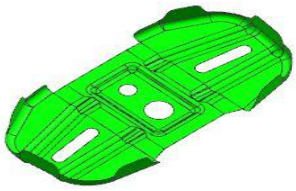

Fig (3)drawing processPartsmodels

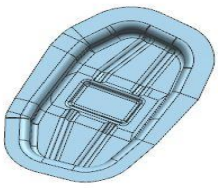

Fig. (4) Addingaddendumin the geometryparts

After adding of the addendum the sidefloorreinforcement plate tool seeFig .(6)and Fig.(7).The geometricalmodel in whichthe maximumcross-sectional dimensionofthe punch169.8mm × 80mm,punchradiusR7mm,die-sectional dimensions of170mm × 80mm,die radiusR8mm.

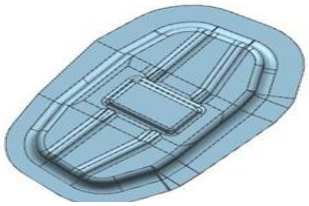

Fig. (6)Die

Fig. (7) Punch

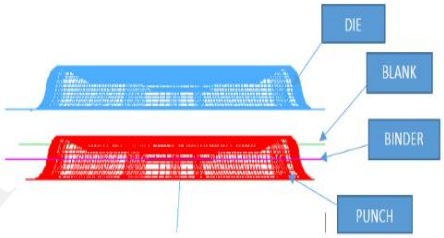

Fig. (8) Geometry tool and blank

Simulation result (FEL analysis)

Among analysis result thickness, strain and FLD (forming limit diagram) are the key results to analysis before a physical tryout can be carried out. The FEA simulation have be conducted in the PC with 2.5GHZ and RAM of 400 GB. Each simulation require about 60 minutes to finish the jobs submitter.The result of FLD analysisin fig (9) The FLD shows that there is no crack existing in the part the majority portion of the part is in the safe area. This indicates that the formability of this part is good [2]; however, other manufacturing aspects needed to be taken into consideration as well. Thinning is utilized to estimate the forming quality through change of thickness. If the thinning ratio is below this range, the stiffness of the part will be weakened, which may cause tearing. The thickness distribution of the drawn part is show in Fig. (10). this value is acceptable in terms of the structural integrity.From manufacturing points of view, the drawn part's perimeter is considered with respect to industrial application.

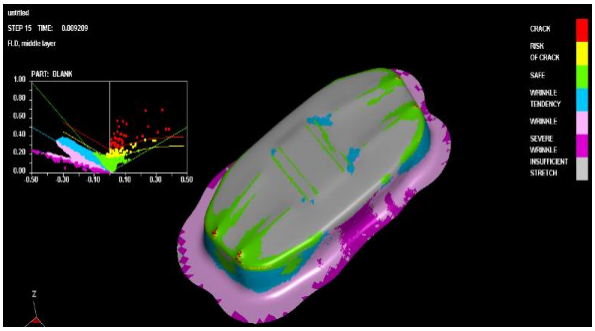

Fig (9) FLDAn analysis result

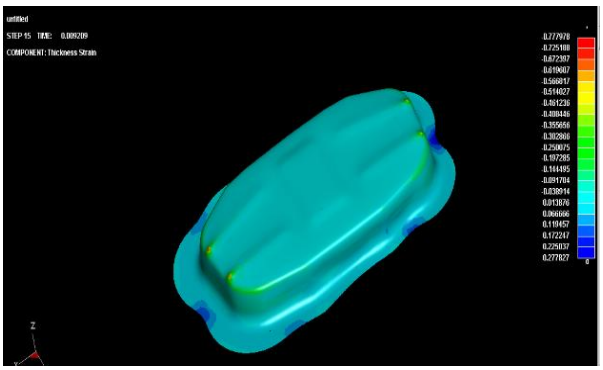

Fig (10) the thickness distribution

CONCLUSION

Computer simulation is very helpful and important for sheet metal forming procedures in the tool design stage, improve design initiative and reduce unnecessary error, obtain appropriate addendum layout, extracting punch and binder by using DYNAFORM software and other processing parameter, reduce cost and time. In this paper the automobile reinforcement plate front side edge beam box studied with the combination of numerical drawing and flanging simulation in the full flow of forming process. The Couse of formation of wrinkles was studied on the basis of metal flow obtained from the simulation the forming limit diagram FLD was also used in conjunction with the finite elements result to predict the automobile reinforcement plate front side edge beam box.The availability of simulation is proved by comparing the simulation result with the experiment.

So finite element analysis (FEA) simulation is very important tool for analyzing complex dimensions .sheet metal forming problems related to potential forming defect such as wrinkling and crack.

REFERENCES

[1] A. Anderson: Comparison of Sheet Metal Forming Simulation and Try-out Tools in Design of Forming Tools, J. of Engineering Design, v. 15. (2004)

[2] eta/DYNAFORM user's manual version 5.7 engineering technology associates inc.1998-2009

[3] A. Anderson: Information Exchange within Tool Design and Sheet Metal Forming, J. of Engineering Design, 2001. v. 12.

[4] J.N. Reddy, an introduction to finite element method, McGraw Hill, Inc., 2nded. 1993

[5] Zhang ying, development of sheet metal forming simulation [J]. Journals of plasticity engineering, dec.1996 .