International Journal of Recent Advances in Multidisciplinary Topics
Volume 2, Issue 4, April 2021
https://www.ijramt.com | ISSN (Online): 2582-7839

Design and Forming Analysis of BIW Sheet Metal Component Using Autoform Simulation Software

Rahul Taware[1*], Vidyadhar Rajmane[2]

[1,2]Department of Mechanical Engineering, Shivnagar Vidhya Prasarak Mandal College of Engineering, Baramati, India

***Abstract*: BIW is a frame member of a vehicle, which forms the chassis of vehicle. It has so many small features with ups and downs. The component is manufactured by draw type die. This is because to reduce the draw depth and it has two rows of draw beads. It seems to be difficult to form due to their small radius fillets. In this analysis, there is a need to find out the forming feasibility of this component and if not possible in single stage then we need to modify the die design with least modification and need to optimize the draw bead profile and draw radius in such a way that we can get the good component in a single stage itself. For thin and very big forming dies, we need to do gravity analysis. If our die face is profiled one, then we may need to do binder wrap analysis. The component wrap analysis results are taken to the forming analysis. After the forming analysis, if trimming is there means just before trimming analysis we have to do mesh Coarsening. Then the coarsened mesh and its results are taken for trimming analysis. The results of the trimming analysis are taken to the spring back analysis. From the results of the spring back analysis, further we can cross check the spring back of the real component. The analysis results obtained is used in sheet metal industry for metal forming process.**

***Keywords*: Analysis, Draw Die, Forming Die, Tonnage, etc.**

1. Introduction

The use of simulation software in metal forming process has increased significantly in recent years as the benefit of troubleshooting and optimizing process on the computer rather than through extensive shop floor trials have been realized. The rapid development of software technology, together with faster and lower cost computer hardware, has recently enabled many manufacturing operations to be modeled cost-effectively that only a few years ago would have been considered impractical. Many of these advances have been made possible by tailoring and optimizing programs for specific applications, which has resulted in the general terms of “sheet forming” and “bulk forming” applied to different types of process modeling software [8-10]. For examples, sheet metal forming is currently developing much interest in the globe as a means of reducing both the development cost of stamping a new part and the production lead-time. These costs are accumulated over the entire development process from initial part design to the final Production tool. The correct software tool will depend on both the application and the stage of product development. Design of sheet metal forming is traditionally relying on the experience accumulated by tool design engineers through long and costly trial and error experiments. Simple empirical methods provide some guidelines for cases similar to those on which these methods were developed. In increasing number of cases with complex geometry's and thinner stronger materials, experiments are used extensively consuming time and money before providing a workable solution. These experiments usually lead to one severity conclusion (the pressed part fails or not) with little if any information on the safety margin. The arising need to produce more complex parts in the most economical way calls for a tool design methodology capable of providing the engineer with a more detailed insight of the physical behavior. This applies especially for parts made of new materials for which experience lacks another strain on the overall design cycle of a new product in the Traditional approach results from the fact that before prototypes for testing can be manufactured a whole series of preliminary tasks must be carried out (sequential engineering). The target of a 'fast-to-the market' release of new products ideally requires a simultaneous engineering approach in which the assessment of manufacturability can be explored as early as possible in the design cycle. These emerging needs called for a new tool design methodology based on numerical simulation. The new CAE methodology investigates and simulates the physical phenomena developing during sheet metal forming. This is achieved by using a numerical simulation tool – AUTO FORM, HYPER FORM, L.S –DYNA, DYNAFORM, PAM-STAMP etc. - which is used throughout a multistep design process-formability evaluation with numerical simulation (FENS) [1-2]. This approach provides the design engineers with a detailed insight or what is happening in sheet metal forming and which are the causes of troubles, thus easing their solutions. Sheet metals parts have the advantage that the material has a high elastic modulus and high yield strength so that the parts produced can be stiff and have a good strength-to- weight ratio. A large number of techniques are used to make sheet metal

---

*Corresponding author: 7rahultaware97@gmail.com parts. Spring back effect is important in forming processes [3].

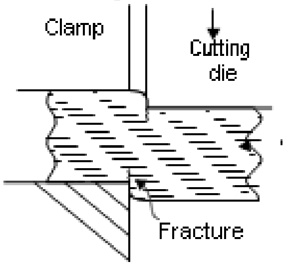


Fig. 1. Magnified section of blanking a sheet showing plastic deformation and cracking

As sheet is usually delivered in large coils, the first operation is to cut the blanks that will be fed into the presses; subsequently there may be further blanking to trim off excess material and pierce holes. The basic cutting process is shown in Figure 1. When examined in detail, it is seen that blanking is a complicated process of plastic shearing and fracture and that the material at the edge is likely to become hardened locally. These effects may cause difficulty in subsequent operations and information on tooling design to reduce problems can be found in the appropriate texts.

A. *Bending*

The simplest forming process is making a straight line bend. Plastic deformation occurs only in the bend region and the material away from the bend is not deformed. If the material lacks ductility, cracking may appear on the outside bend surface, but the greatest difficulty is usually to obtain an accurate and repeatable bend angle. Elastic spring back is appreciable [13-14].

In roll forming machines, there are a number of sets of rolls that incrementally bend the sheet, and wide panels such as roofing sheet or complicated Clamp die Fracture channel sections can be made in this process. A technique for bending at the edge of a stamped part is flanging or wiping as shown in Figure. The part is clamped on the left-hand side and the flanging tool moves downwards to form the bend. Similar tooling is used is successive processes to bend the sheet back on itself to form a hem. If the bend is not along a straight line, or the sheet is not flat, plastic deformation occurs not only at the bend, but also in the adjoining sheet. If the part is curved near the flange or if both the flange and the part are curved, as in figure, the flange may be either stretched or compressed and some geometric analysis is needed to determine this.

B. *Section Bending*

In Figure 2, a more complicated shape is bent. At the left-hand end of the part, the flange of the channel is stretched and may split, and the height of the leg, h, will decrease. When the flange is on the inside, as on the right, wrinkling is possible and the flange height will increase.

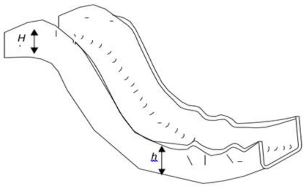


Fig. 2. Inside and outside bends in a channel section

2. Theory

The stamping product cycle involves sheet metal forming operations. The forming operation leads to induced residual stresses. There are chances of split, wrinkle and thinning on panel by residual stresses. The problem arises due to the split, wrinkle, thinning, which directly affect the formability of component and results obtained are out of acceptability limit In this paper design and analyze of BIW component by using Auto form Simulation software and design die and punch forming surfaces for defect free part using CATIA V5 is discussed. The appropriate capacity press can be selected by knowing the drawing load. Working with the presses of higher capacities may lead to many types of defects such as cracks and tearing. Blank holder pressure needs to optimize over a given range for optimized geometry. The coefficient of friction needs to be optimized for the new geometry. Generally the deep drawing objects are analyzed for their strength and failures with circle grid analysis, which is practically carried out on a sample piece, which is known as formability analysis. Alternatively, the actual trials performed over the component would directly reflect over the ease of `drawing operation’ offered for the said Die design.

3. Design of BIW Component

A. *Engine Mounting Bracket*

Engine mounts have an important function of containing firmly the power-train components of a vehicle. Correct geometry and positioning of the mount brackets on the chassis ensures a good ride quality and performance. As an FSAE car intends to be a high performance vehicle, the brackets on the frame that support the engine undergo high static and dynamic stresses as well as huge amount of vibrations. Hence, dissipating the vibrational energy and keeping the stresses under a predetermined level of safety should be achieved by careful designing and analysis of the mount brackets. Keeping this in mind the current paper discusses the modeling, Finite Element Analysis, Modal analysis and mass optimization of engine mount brackets for a FSAE car. As the brackets tend to undergo continuous vibrations and varying stresses, the fatigue strength and durability calculations also have been done to ensure engine safety.

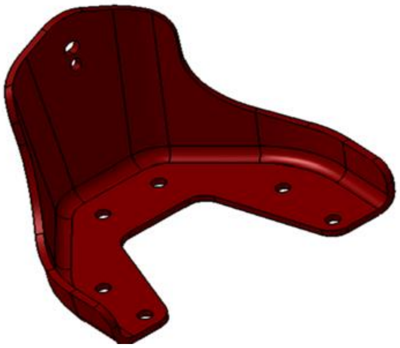


Fig. 3. 3D View of Component

B. *Part Significance*

Table 1 Part significance

Drawing/Part No.	**2855 2620 8211**
Drawing/Part Designation	BRACKET (TRUCK)
Material Thickness	6 mm Thickness
Material Specification	HR-E 46 (BSK-46)
Envelope Dimensions (mm)	188.31 x 224.11 x 247.0
Final mass in kg.	2.948 kg.
Customer Name	TATA MOTORS LIMITED ERC - PUNE

1) *Part Drawing*

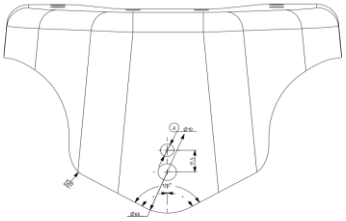


Fig. 4. Front view

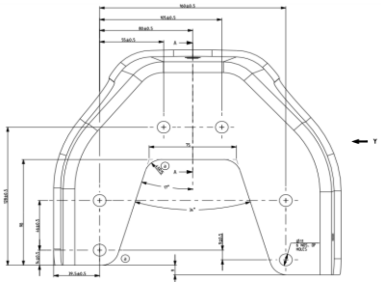


Fig. 5. Top view

Fig 4 and fig 5 shows front nad top view of component.

C. *Material Properties of BSK-46*

BSK- BIG STEEL KEG Tank type geometry used in auto component. (BSK 46)- can be produced by controlled cooling after hot rolling, has composition as under:

Carbon Equivalent (CE):- 0.41 Max

WHERE, C.E.=(C+(Mn/6)+((Cr+Mo+V)/5)+ ((Cu+Ni)/15)) Trace elements other than specified should not exceed 0.2% in total by weight

- Material Tensile strength – 500-640 MPa
- Yield strength – 460-560 MPa
- Poison Ratio - 0.31
- Young's Modulus - 210 GPa.

It is clear from the properties of materials mentioned in table1, that BSK 46 has more tensile, yield and modulus value than the other materials in the list.

Bending Force Calculations

Cutting force calculation:

$\text{Shear strength } (\text{N/mm}^2) = 455\ \text{N/mm}^2$

$\text{Cuttingforce} = \text{Perimeter of part (mm)} \times$

$\text{Thickness of part (mm)} \ \times \text{Shear strength } (\text{N/mm}^2)$

$= 512.3 \times 6 \ \times 455 \ = 1398579\ \text{N}$

$= 1398.579\ \text{KN}$

$= 142.56\ \text{TONNES}$

Bending force calculations:

Assuming bending force 80% of cutting force.

$\text{Therefore Bending Force} = \frac{142.56 \text{ x } 70}{100}$

$\text{Bending Force} \ = \ 99.79\ \text{Tonnes}$

Therefore, bending force for 1 single part is 100 tonnes.

Similarly, for 2 parts bending force will be,

$\text{Bending Force} \ = 2 \text{ x } 99.79$

$\text{Bending Force} \ = 199.58\ \text{Tonnes}$

Here consider joining part off material 10%

Then, $\text{Bending Force at joint} \ = \frac{199.58 \times 10}{100}$

$= 19.95\ \text{Tonnes}$

Therefore, Total Bending Force = Bending Force for two parts + Bending force at joint

$= 199.58 \ + \ 19.95$

$= 219.53\ \text{Tonnes}$

Hence total Bending force for Engine Mounting Bracket is 219.53 Tonnes.

D. *Blanking Tool Calculations*

1) *Cutting Clearance for Blanking Tool*

Cutting Clearance for 6 mm BSK – 46 material is calculated as follows; For 6 mm thick material, Cutting Clearance (both side) = 15% of thickness (From standard chart) = 0.15 x 6 =0.9mm. Therefore cutting clearance for Die is 0.9 mm for both side i.e. 0.45 mm per side clearance. OP50

2) *Actual Tonnage Calculation for Blanking Tool Material thickness = 6 mm*

Total Perimeter of Blank (mm) = 1207.1 mm (From CAD model)

$\text{Shear strength } (\text{N/mm}^2) = 455\ \text{N/mm}^2$

$\text{Cuttingforce} = \text{Perimeter of part (mm)} \times$

$\text{Thickness of part (mm)} \ \times \text{Shear strength } (\text{N/mm}^2)$

$= 1058.3 \times 6 \ \times 455 = 2889159\text{N}$

$= \frac{2889.159}{9.81}\ \text{KG}$

$= 294.51\ \text{TONNES}$

F = 294.51 Tonnes ≤ 300 Tonnes

Therefore, Schuler - 300T Press is selected for Blanking tool.

4. Simulation of BIW Component

Tooling sequence for Engine mounting bracket has major influence on part forming. Therefore we considered following tool sequence for bracket.

Table. 2 Operation sequence

COMPONENT PROCESS SEQUENCE	
OP05	Blanking
OP10	Forming 1
OP20	Forming 2
OP30	Piercing (6 Station) & Cam Piercing (2 Station)
OP40	Top Windo Cut & Cam Piercing (2 Station)
OP50	Part Off & Piercing (6 Station)

OP05 – Blanking Tool

A blanking die produces a flat piece of material by cutting the desired shape in one operation. The finished part is referred to as a blank. Generally a blanking die may only cut the outside contour of a part, often used for parts with no internal features. It is not practically possible to manufacture single part in forming therefore we have taken two parts as per feasibility of manufacturing for Engine mounting bracket. If we take single part in forming (draw) die it will cause failure of part.

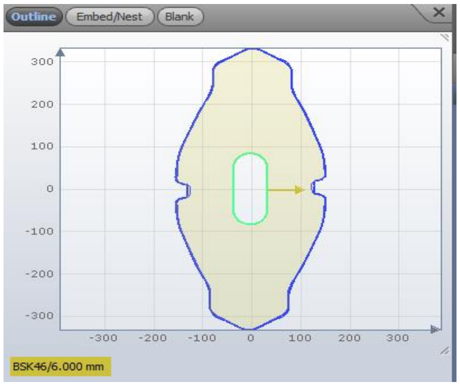


Fig. 6. Development blank in autoform software

Fig. 6 shows blank developed by Autoform software. Autoform software interface requires 3D stp file or igs file for capturing surface of part. After it convert it into development blank. In Engine mounting bracket as we considered two part for balance forming, it gives us above development blank.

A. OP10 – Forming-1 Tool

In Engine mounting bracket it is not practically feasible to obtain final part in single forming tool, therefore we have considered two forming operations Forming-1 tool and Forming-2 tool. Deep drawing operation is a sheet metal forming process in which a sheet metal blank is radially drawn into a forming die by the mechanical action of a punch. It is thus a shape transformation process with material retention.

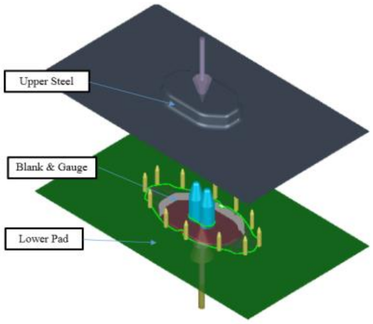


Fig. 7. Forming-1 Tool setup

The process is considered "deep" drawing when the depth of the drawn part exceeds its diameter. This is achieved by redrawing the part through a series of dies. The flange region (sheet metal in the die shoulder area) experiences a radial drawing stress and a tangential compressive stress due to the material retention property. These compressive stresses (hoop stresses) result in flange wrinkles (wrinkles of the first order). Wrinkles can be prevented by using a blank holder, the function of which is to facilitate controlled material flow into the die radius.

B. OP20 – Forming-2 Tool

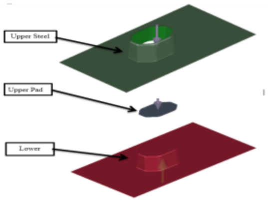


Fig. 8. Forming-2 Tool setup

Part is partially formed in Forming Tool-1, and its final shape is formed in second forming tool. The setup for OP20-Forming Tool is as shown in fig. In forming tool-2 part is situated upside down as shown in fig. 8. This upside down position of part is preferable for ejection. Forming tool-2 consist of ejection pad in upper assembly of die. Upper pad is movable part and its movement is controlled by cution pins of press bed. Firstly pad holds the part in the start of operation in position, and similarly at end of operation i.e. after hitting the part it eject the part.

C. OP-30 Piercing (6 Station) & Cam Piercing (2 Station)

1) Before Piercing

2) After Piercing

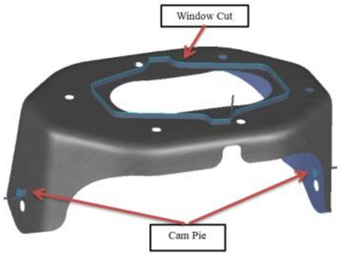
Fig. 9. Pricing tool setup

P-30 is cutting tool which consist of two stations, one vertical piercing and another horizontal cam piercing. Vertical piercing done at 6 positions and horizontal cam piercing at two positions as shown in fig. 11. Station-1 consist of six piercing holes of 11mm diameter and punching is done by vertical punch. Station-2 consist of two whole piercing of 12mm diameter. At station-2 horn piercing done by two cam which are opposite to each other

D. OP-40 Top Windo Cut & Cam Pie (2 Station)

Op-40 consist of two stations, top window cutting and Cam piercing, The shape of this top window cannot be defined in blanking die before forming predations as it get stretched in in forming, therefore it had done after forming the parts. Top window cutting done as per final part dimension required. At station-2 second hole of 10 mm diameter is pierced by two cams, which are positioned opposite to each other. As we can see in drawing there is only 17.5 mm center distance in part flange, therefore it is not feasible to pierce both holes in same cutting tool.

1) Before Piercing



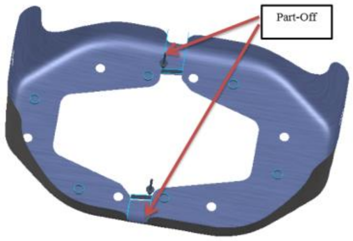
2) After piercing

Fig. 10. Top Windo Cut & Cam Pie (2 Station)

E. OP-50 Part Off and Piercing

Part off and piercing is two station die, which pierce 6 holes of 11 mm diameter at first station. And at second station it part off into two separate two parts. Part off operation done simultaneously with the help of two separate punches and die sets.

1) Before Pierce and Part –off



2) After Piercing and Part off

Fig. 11. Top Windo Cut & Cam Pie (2 Station)

5. Manufacturing and Testing

After deciding Operational sequence, tools has been designed as per sequence. Designing of press tools has done by CATIA V5 R26 as per manufacturing feasibility. After designing DAP (Design Approval Process) has been done with customer. DAP is most important in tool designing and it cannot go ahead without customer DAP. After approving we released manufacturing 3D and 2D on shop floor.

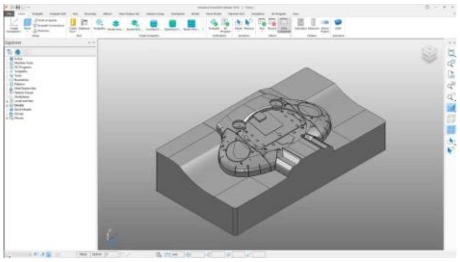
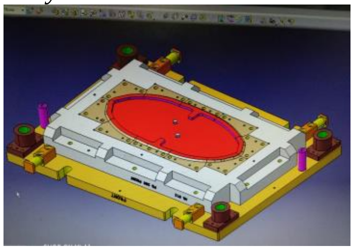


Fig. 12. NC Programming in Delcam

3D model from CATIA V5 cannot be machined as it in the ".catpart/.cat product format, therefore it needs to be converted into stp file format. Step file format is readable with any 3D CAD or CAM software. Manufacturing programming of tool's parts has done using Delcam software, and program has given to Numerical control of VMC (Vertical Machining Centre) machine. Machining of model is always done with tool which is selected in programing. After machining parts are assembled in two main assemblies i.e. Upper assembly and Lower assembly. After matching punch and die block are sent outside for vacuum hardening. Punch and die block are always hardened in the range of 60-65 HRC (Hardness Rockwell Constant). After assembling final inspection of tools has done. Then it is taken in press shop for trial.

A. Testing of Blanking Tool

1) Lower Assembly



2) Upper Assembly

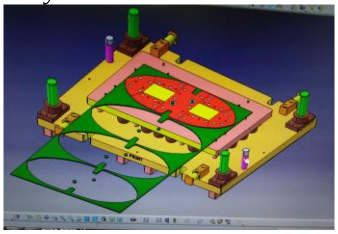


Fig. 13. CAD Model of OP05 – Blanking Tool

Fig. 14. Actual Blank – Product of Blanking Tool

Tryout is the step in the engineering and manufacturing process of a tool when the tool is first loaded into the press in an initial attempt to produce a part. Next, the tool undergoes extensive fine tuning during tryout, which is a cost and time intensive step on the path to the successful production of quality tool. As correctional work and modifications are inevitable, every correction loop that can be avoided offers an immediate advantage in terms of time and money. However there are always some variation in part dimension in forming tools. The Try-out team must carry out several correction loops on the tool until the tool can be used to produce a part of the required quality.

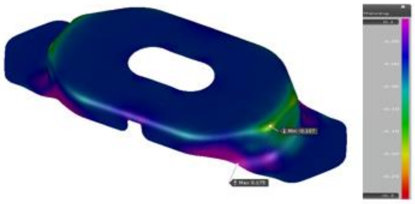
6. Results and Discussion

In metal forming simulation, the forming of sheet metal is simulated on the computer with the help of special software. Simulation makes it possible to detect errors and problems, such as wrinkles or splits in parts, on the computer at an early stage in forming. In this way, it is not necessary to produce real tools to run practical tests. Forming simulation has become established in the automotive industry since it is used to develop and optimize every sheet metal part. To illustrate the metal forming process, there must be a model of the real process. This is calculated in the software using the finite element method based on implicit or explicit incremental techniques. The parameters of the model must describe the real process as accurately as possible so that the results of the simulation are realistic. The metal forming simulation enables fast and accurate simulation of the entire forming process including drawing and secondary operations as well as spring back. In this way, the part can be developed completely and efficiently. The typical parameters for forming simulation are, for example, part and tool geometry, material properties, press forces and friction. The simulation calculates stresses and strains during the forming process. In addition, simulations allow for the recognition of errors and problems (e.g. wrinkles or splits) as well as results (e.g. strength and material thinning). Even spring back, the elastic behavior of material after forming, can be predicted in advance. Forming simulation also provides valuable information about the influence of process variations on stamping robustness.

A. Software Result: OP-10 Forming-1 Tool

1) Thinning

Maximum thickness is +0.175 which is occurred at flange side of part. Minimum material thickness is -0.187 which is observed at deep draw section of part as shown in fig. below.



2) Formability

Fig shows formability analysis of component, it is observed component is supposed to thicken at the radius portion simultaneously it tends to compression at the middle of flange.

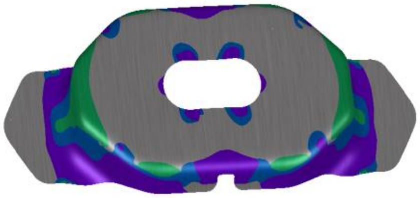
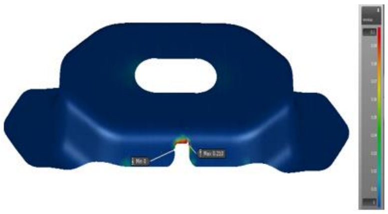


Fig. 15. Analysis of sheet metal using Autoform CAE Software

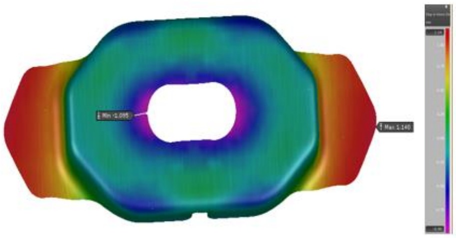
3) Wrinkles

Fig shows wrinkle analysis of forming – 1 tool. There is only one portion where forming of wrinkle occurs, at the joint of two parts in the component as we can see in the fig. The maximum magnitude of wrinkles is 0.21 mm, however it doesn't affect the component as its scrap in part off tool.



4) Spring back

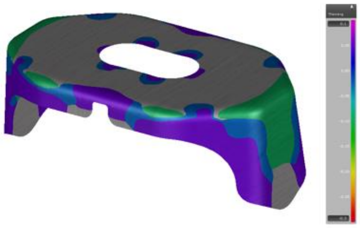
In forming tool 1, there is more spring back effect in outer edge of the part as shown in below fig. At inner cut out section it is -1.095 i.e negative therefore it tends to vibration.



B. OP-20 Forming-2 Tool

1) Thinning

In forming-2 tool maximum thinning occurs at the flange of the component as shown in the fig. As compared to formong-1 tool thinning is less in forming-2 tool.



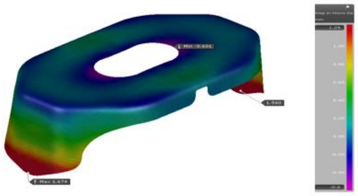
2) Formability

Figure shows formability analysis of Forming-2 Tool. Thickening of component continues till the end of forming-2 tool at the side flange as shown in figure. There is no any excess thinning observed at the job.

3) Wrinkles

In forming-2 tool wrinkles are observed at two portions, at the joint of two parts which are formed in forming-1 tool and at the concave shaped flange as shown in fig.

4) Spring back

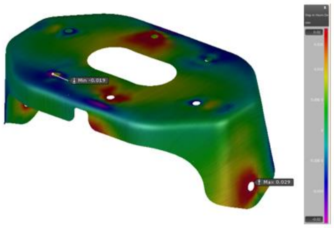


Spring back analysis of forming-2 tool is as shown in fig. Its magnitude is higher at extreme point on flange of component. Higher magnitude observed of spring back effect in the component is +1.674. Similarly minimum magnitude of spring back effect is observed of -0.601 at internal cutout section as showed in figure below.

C. OP-30 Piercing 6 Stations & Cam Pierce

1) Spring back

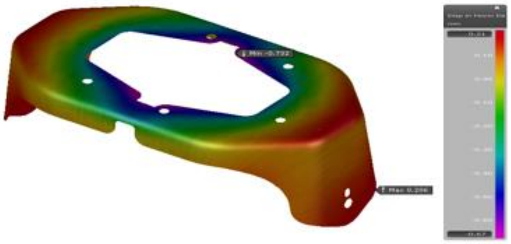
Spring back effect is also observed in Piercing 6 station and cam piercing die. The maximum magnitude of springbuck is observed at horizontal cam piercing hole of +0.029. There is also medium spring back effect observed at middle cross holes.



D. OP-40 Top Window Cut and Cam Piercing

1) Spring back

In top window cutting operation maximum spring back is +0.206 which is observed at outer edge of flange and minimum spring back is -0.732 at upper wall of part as it is held in position by stripper plate. The force required in window cutting operation is large as compared other piercing operation as it cutting perimeter is large than others.



2) OP Part Off and Piercing

Spring back effect in part off tool is shown in figure below. The maximum spring back effect is + 0.070 which found at extreme point on lower side of flange and minimum spring back is – 0.223 which found at upper wall near cutting edge of component. However spring back effect in in part of tool is small as compared to other tools.

E. Forces Developed

1) In Forming-1 Tool

The forces developed in upper, lower die and pad assembly of Forming-1 tool of the BIW component is as follows

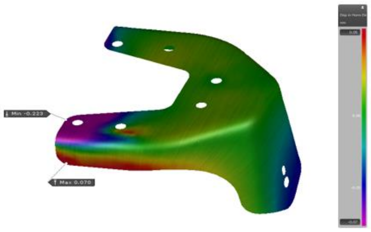
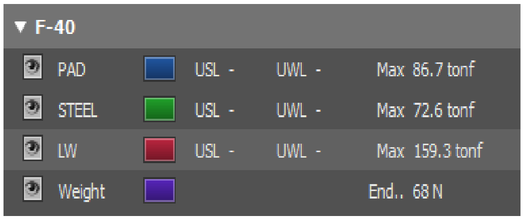


Fig. 16. Forces Developed In Forming-1 Tool

2) In Forming-1 Tool

The forces developed in upper, lower die and pad assembly of Forming-2 tool of the BIW component is as follows



▼ F-40

PAD	🟦	USL -	UWL -	Max 86.7 tonf
STEEL	🟩	USL -	UWL -	Max 72.6 tonf
LW	🟥	USL -	UWL -	Max 159.3 tonf
Weight	🟪			End.. 68 N

Fig. 17. Forces Developed In Forming-2 Tool

7. Conclusion

The BIW component is to be analyzed for finding the forming feasibility and it is formable in a single stage. On this process, we could be able to optimize the die design, draw radius, blank holding force, draw bead profile etc. For thin and very big forming dies we need to do gravity analysis. If our die faces in profiled one then we may need to do binder wrap analysis. The coarsened mesh and its results are taken for trimming analysis. The results of the trimming analysis are taken to the spring back analysis. From the results of the spring back analysis, we can cross check the spring back of the real component. Thinning and Formability are ascertained in this study. In order to expand the range of application of the developed method, parts with more complex geometries can be considered as future scope of work.

References

[1] G.H. Farrahia, A. Ahmadia, K. Reza Kasyzadeh, “Simulation of vehicle body spot weld failures due to fatigue by considering road roughness and vehicle velocity” Department of Mechanical and Instrumental Engineering, Academy of Engineering, Peoples’ Friendship University of Russia, 7 August 2020.

[2] Prafull Prakash Lolge, M. R. Dhanvijay, “Stamping Process Simulation With The Application of Different Material And Processes to The Same Panel Using Autoform CAE Tool”, ELK Asia Pacific *Journal Of Manufacturing Science And Engineering,* vol. 3, no. 2, 2018.

[3] Qin Huan, Guo Yi, Liu Zijian, Liu Yu, “Shape optimization of automotive body frame using an improved genetic algorithm optimizer” State Key Laboratory of Advanced Design and Manufacturing for Vehicle Body, College of Mechanical and Vehicle Engineering, March 2018.

[4] B Kedar, B Ashok, Nisha Rastogi, “Design and Analysis Of Automobile Components Using Industrial Procedures” IOP Conf. Series: Materials Science and Engineering 2017.

[5] F. Greco, E. Deckers, J. Stroobants, S. Van Poppel, W. Desmet, “Finite element simulation of the dynamic behaviour of deep drawn components with accurate thickness description” Department of Mechanical Engineering, Division PMA, Celestijnenlan, Belgium, October 2017.

[6] Sanjay Kumar S.M., D.P.Girish, “Design and Formability Analysis of Fender Component”, IJRET: International Journal of Research in Engineering and Technology, vol. 5, no. 4, May-2016.

[7] Fredrik Henriksson, Kerstin Johansen, “On Material Substitution in Automotive BIWs – From Steel to Aluminum Body Sides”, Division of Machine Design, Linköping University, Sweden, *Elsevier* B.V., May 2016.

[8] Dae Up Kim, Young Choi, Bong Yong Kang, “A Study on Process Design of Automobile Parts Using Extruded Material by Die Forming” Hindawi Publishing Corporation Advances in Mechanical Engineering, 2014.

[9] Ulhas K Annigeri, Y P Deepthi, Raghavendra Ravi Kiran K, “Design, Development And Analysis Of Forming Tool For Side Panel Of An Automobile” *International Journal of Mechanical And Production Engineering*, vol. 2, no. 6, June-2014.

[10] Sachin Ramdas Jadhav, Sunil Hiraman More, Swapnil S. Kulkarni, “Die Design for Formed Component using inputs from FEA for determining the most suited values for the Design or Process parameter", International Journal of Advanced Engineering Research and Studies, pp.95-98 April,2014.

[11] Mr.Amit ,D.Madake, Dr.Vinayak, R.Naik Mr.Swapnil, S.Kulkarni, "Development of a Sheet Metal Component with a Forming Die Using CAE Software Tools (Hyper form) For Design Validation and Improvement", *International Journal of Modern Engineering Research* (IJMER) , vol. 3, no. 3, pp-1787-1791, May- June, 2013.

[12] A. Al-Zaher, W. ElMaraghy, Z.J. Pasek, “RMS design methodology for automotive framing systems BIW”, Industrial and Manufacturing Systems Engineering, University of Windsor, Canada, May 2013.

[13] Nagur Aziz Kamal Bashah, Norhamidi Muhamad, Baba Md Deros, Ahmad Zakaria, Shaharum Ashari, Achmed Mobin, Department of Mechanical and Materials Engineering, Faculty of Engineering and Build Environment, Universiti Kebangsaan Malaysia, October 2012.

[14] Mohammad Ali Farsi, Behrooz Arezoo, “Bending Force and Spring-Back in VDie-Bending of Perforated SheetMetal Components” *Journal of the Brazil Society of Mechanical Science & Engineering*, March 2011.

[15] D. Mundoa, R. Hadjit b, S. Donders b, M. Brughmans, “Simplified modelling of joints and beam-like structures for BIW optimization in a concept phase of the vehicle design process” Department of Mechanical Engineering, University of Calabria, 87036 Arcavacata di Rende, Italy, February 2009.

[16] Mehmet Firat, Bilgin Kaftanoglub, Orhan Eser, “Sheet metal forming analyses with an emphasis on the springback deformation” The University of Sakarya, Department of Mechanical Engineering, 54187 Adapazari, Turkey May 2007.

[17] M. Grujicic, V. Sellappana, G. Arakere, Norbert Seyr, Marc Erdmann, “Computational Feasibility analysis of direct-adhesion polymer-to-metal hybrid technology for load-bearing body-in-white structural components”, BMW Group, Research and Technology, Knorrstraße 147, 80788 Munchen, Germany May 2007.

[18] S. A1 Azraq, R. Teti, J. Costa, “Springback prediction with FEM analysis of advanced high strength steel stamping process” Dept. of Materials and Production Engineering, University of Naples Federico II, Naples, Italy 2006.

[19] Z. Marciniak, J.L. Duncan, S.J. Hu, “Mechanics of Sheet Metal Forming” An imprint of Elsevier Science Linacre House, Jordan Hill, Oxford Butterworth-Heinemann Publication, Second edition, 2002.