ELK ASIA PACIFIC JOURNAL OF MANUFACTURING SCIENCE AND ENGINEERING
ISSN Online: 2394-0425; Volume 3 Issue 2 (2018)

ELK
Asia Pacific Journals
www.elkjournals.com

STAMPING PROCESS SIMULATION WITH THE APPLICATION OF DIFFERENT MATERIAL AND PROCESSES TO THE SAME PANEL USING AUTOFORM CAE TOOL

PRAFULL PRAKASH LOLGE
T.Y.B. Tech Student,
Department of Production Engineering and Industrial Management,
College of Engineering, Pune, Maharashtra, India.
lolgepp16.prod@coep.ac.in

M. R. DHANVIJAY
Asst. Professor
Department of Production Engineering and Industrial Management,
College of Engineering, Pune, Maharashtra, India.
mrd.prod@coep.ac.in

ABSTRACT

The Aerospace and Automotive industry has increased use of simulation technology to achieve high standard quality product economically in less time. The simulation technology defined the results of panel after forming operations. This Paper highlights the analysis of result of simulation for different material over the same panel and analysis of parameter and process which leads to development of panel to prove the component area in acceptable formability limit. The development of panel will consider manufacturing and assembly feasibility by design intent. The simulation results will be obtained by CAE software used in designing forming tool. This Technology helps in reducing the whole part development process as compared to what happen with conventional method. The Part development need to improve without any quality issue by lesser effort and easy to setup the whole model and other feature with effective design parameter. Though the main is to develop a sheet metal component by designing sheet metal die effectively by saving time and cost, we need to identify the problems through simulation results by CAE software and based on the analysis prepare the changes and modification in profile and make revisions changes in the respective part data and part packaging module. Finally, work output develops a feasible and effective products and dies. Thus, by use of CAE software, design of dies and products is done economically because the changes in design, modification are solved in engineering stage with applying a combination of effective parameter. Otherwise without use of this technology, the die manufacturing process from design to try-out could cost more and complicated one. The final stamping forming simulation results based on the finite element method and it aims in material optimization, improved product quality, reduced product development cycle and reduced manufacturing cost by proving panel with respect to actual by 90%.

Keywords: CAE software, Design intent, Finite element method, Formability limit, Simulation technology etc.

1. INTRODUCTION

In today's world, each and every component of the vehicle which is mounted on the BIW fixture is manufactured by sheet metal forming process. The main advantages of this process are less cost and effective material utilization for the manufacturing industries. This process added with the tool design, product development, CAE, CAM tools provides benefits in continuous improvement in tolerance range and mechanical properties of sheet metal parts. Due to the application of CAE and CAM tools, exact shape of the part with respect to cad data is formed by metal forming process. This saves not only manufacturing operations but also manufacturing cost. Consequently, the introduction of CAE and CAM tools to forming process is a complex task. [1, 2, 3]

The tooling industries which manufacture the automotive sheet metal components require CAE tool besides skill, standard and experience which are utilized in the machine shop. The use of CAE tool is increasing continuously due to the cost saving of the product cycle. Thus the application of CAE technique has become effective in automotive industry. [4]

The product development cycle follows the process: product design, simulation using CAE tool, definition of process, die design, CAM and assembly. The CAE technology is most widely used in crashform, drawing, forming and all other operations which are involved in the sheet metal part production. This plays critical role in deciding panel quality, reducing number of operations and utilization of effective parameter. In product development cycle, if we directly jump from product design to die design and neglect use of CAE tool then it may involve high modification cost of panel due to non-feasible changes in the panel. The CAE tool is introduced in product cycle by two ways: one is experience based technology and the other is use of software technology by achieving simulation results within acceptable formability limits. The former deals with the advanced theory knowledge and empirical formulae which will decide the output i.e. simulation result. The experience technology has advantage that it can determine the material flow easily and disadvantage that determination of results of the part within acceptable formability limit is not possible. [1,2]

The paper contains validation of the simulation results with respect to actual results. It contains terminology and descriptions of the simulation parameters. This validation has been done by the use of CAE tool i.e. Autoform R7 plus. The analyses have been done on simulation results with variation in parameter and process. The comparison between the simulated results using different sheet materials of various steel grades are being analyzed to achieve better result by drawing operation. The crashform operation is introduced in the current work. The difference between drawing and crashform operations is illustrated. In this paper, the output of simulation result is matched with actual results by 90% for Bracket Heat Shield and Simulation results are achieved within acceptable formability limit by crashform operation to save cost and time by 30% with respect to drawing operation for Front Roof Bow panel.

1.1 OVERVIEW

1.1.1Problem definition

The product cycle involves sheet metal forming operation. 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 [4].

1.1.2 Need to solve the problem

In tooling industry, the panel comes from styling to product design and product design team works on the features of part. They make features of the part such that it is feasible for manufacturing by using one step simulation. The output part given by product design team goes to the CAE team, where feasibility of part is checked and depending on the result, optimization of parameters is done to obtain results within the acceptable limits. If some part features are not feasible then CAE team communicates with product design team and changes are made. Thus, this cycle continues until the panel result is within acceptable limits. If CAE tool is not used, then the effect is visible in the try-out stage, where results do not lie within the acceptable limits. In this case, rework needs to be done in the product cycle. The rework can be change in features of the die or it can be changing the die. Finally, it will affect final cost of die [4].

1.1.3 Problem solution

The simulation of the part data is done by using the CAE tool. It is the best way to get results within the acceptable formability limits. The simulation process involves various stamping processes. The cad data optimized in modelling software needs to be assigned in the simulation software by different stamping processes in sequence. The cad data used in CAE tool as reference. In simulation, the parameters need to be assigned to each stamping process with the use of standardize data. The simulation involves iteration. The parameters are varied to get the results of simulation process within the acceptable formability limits. This will prove the panel in actual with respect to cad data. It involves trial and error time as per experience level. The simulation involves optimisation of the parameters. The parameters optimisation depends upon the panel and stamping process. As parameters are optimised, the quality of the panel improves.

1.1.4 Objective and scope of the work

1. Check the manufacturing feasibility of a panel.
2. Study the simulation result by using different parameter on panel.
3. Optimize the simulation parameter.
4. The scope of paper to determine the feasibility of panel with different methodology.
5. It will determine the behaviour of different parameter and optimize the parameter.

2. VALIDATION OF ONE STEP SIMULATION APPROACH TO FORMING PROCESS

Here a simple panel takes to analyse the forming process. The component name is BRACKET HEATSHIELD which is used in two wheeler vehicle. The validation of sheet metal panel is to solve the problems physically. For that, 1st stage is to develop dieface in CATIA V5/any modelling software. Then the designed panel in CATIA V5/any modelling software used as reference face in simulation software. Following operations are required to complete the panel as per analysis.

1. Blanking
2. Forming
3. Piercing

The result obtained after simulation process compared with the actual manufactured component. It shows variation in thickness after manufacturing.

2.1 Objectives of the Case study

1. Analysis of simulation of manufactured component.
2. Comparison of variation in thickness between results of actual and simulation.

ISSN Online: 2394-0425; Volume 3 Issue 2 (2018)

2.2 Modelling of sheet metal component

Product designer developed a cad model. The part model created as solid model in CATIA V5. But the CAE tool requires surface model for simulation purpose and to predict results. While modelling in product design, predict one side of surface as master and as per consideration of master other features are added by adding thickness to master surface. Master is referred to as the surface on which thickness has implemented. Modelling is done to enable material flow in proper way to better results. The cad model of component is as shown in figure. **(Refer Figure 1)**

2.2.1 Part Details

Part Name -BRACKET HEAT SHIELD.

Material - D513

Sheet Thickness - 2mm.

2.2.2 Part Drawing

Isometric representation and 2D representation of the Bracket Heat shield is shown in figure. **(Refer Figure 2)**

2.2.3 Dieface generation

Generated dieface for the Part data i.e. Bracket Heat Shield is as shown in figure. The main aim behind dieface making is to make panel as smooth as to enable the flow of material inward by the application of bead, Punch and die radius, binder surface. The dieface is used in Crashform/drawing/forming operation as reference surface in simulation to achieve results. The following are basic steps to make dieface. **(Refer Figure 3)**

1. Extension
2. Development
3. Binder development
4. Addendum
5. Development
6. Cut off
7. Merge with Model
8. Draw Panel.

2.3 Simulation Analysis

Here, the dieface made in CATIA V5 is sent to Autoform (CAE) software to achieve simulation results. In Autoform, the sequence of operation defined in a proper way with optimised parameter to each operation. The blank also plays a critical role in forming operation with its parameter like roll angle and material etc. The draw beads used in drawing operation or physical bead can also make at there to control the flow of material, whereas beads are not defined in crashform operation. The results of drawing process for the Bracket Heat Shield in simulation are as follows.

2.3.1 Formability

The formability means the strain state in the strain based forming limit diagram as shown in figure. The results are highlighted with different colours. The representation of colours with meaning is as follows. **(Refer Figure 4 and 5).**

1. Split (Red)
2. Risk of split (yellow)
3. Excessive Thinning (orange)
4. Safe (green)
5. Compression (blue)
6. Insufficient stretch (Grey)
7. Thickening (Purple)

Here by formability, the simulation shows insufficient stretch and safe area. As this is Inner part so in this case, insufficient stretch is acceptable and rest area is safe so this part is proved as per requirement.

2.3.2 Thinning

The thinning shows the thinning or thickening of the sheet metal with respect to original thickness. The thinning considers only mechanical strain it doesn't consider the thermal influence. The thinning for the panel is as shown in figure. The thinning is calculated by using following formulae.

"Thinning = (current thickness- Initial Thickness)/Initial thickness"
(1)

Thus, the thinning show change in thickness after simulation, it needs to keep within limit by enabling flow of material as per requirement. **(Refer Figure 6 and 7)**

2.3.3 Wrinkle

Wrinkle shows the wrinkling behavior of sheet metal depends on geometric shape of panel. The simulation software shows the wrinkling behavior of sheet metal as per color coding scale defined by user. Mostly wrinkle occurs at concave areas. All the concave areas won't be wrinkle. If tool contact is there, then it can be part feature. If there is no tool contact at concave, then it can be wrinkle. **(Refer Figure 8)**

"Wrinkle = C * (T/2)"
(2)where,
T is sheet thickness
C = (1/Radius of convex)

This formula gives strain on the Panel. The wrinkle will be more if width of concave shape is less. Here, the wrinkle for values for Bracket Heat Shield is 0.00 mm so, the part is in acceptable condition. **(Refer Figure 9)**

2.4 Result Comparison

The simulation result and actual result vary slightly as shown in figure. The software package gives 90% matching to actual parts. Factors be like lubrication coefficient, press condition due to which actual result may vary. This can overcome by use of optimize parameter while manufacturing like roll angle, material etc. Here, simulation process has validated. **(Refer Figure 10).**

3. ANALYSIS OF SIMULATION RESULT WITH VARIATION IN PARAMETER AND PROCESS

As per above validation, the change in forming parameter will affect on the panel quality and performance. Here, an automobile part will be taken to study the simulation result by use of standard parameter. Then study will also be done by varying the input parameter like material to compare the results. It will help to achieve effective parameter and comparison between the materials of different grade values for the same panel. The result of simulation will compare all these parameters.

3.1 Development of Dieface

Part name - Front Roof Bow

Material - EDD513 (Extra Deep Draw)

Sheet thickness - 0.7 mm

Development for the dieface has done by CATIA V5/ modelling software. The development of the Front Roof Bow is as shown in figure.EDD is cold rolled low carbon steel. With the use of standardized parameter, the chances of better results increase. **(Refer Figure 11 and 12).**

3.2 Process Planning

The panel has the planned process by its geometry feature. The same process plan needs to plan in Autoform (CAE tool). **(Refer Figure 13).**

OP 10 – Blanking

OP 20 – Forming

OP 30 – Trimming, Piercing, Cam Piercing, Restrike.

3.3 Selection of material

Here, the generated dieface simulated for different material in Autoform R7 plus. The obtained results vary for different material due to material properties. The use of different material for the same panel cannot be done on actual basis which will increase tool cost. So it is effective to simulate the trial and optimize the parameter.

Following material taken for analysis.

1. EDD513 (tensile strength = 165 N/mm^2)
2. IFHS400 (tensile strength =265 N/mm^2)
3. BSK46 (tensile strength = 460 N/mm^2)

3.4 Simulation Analysis

As shown in validation, the feasibility of component won't get by initial parameter. Here analysis is going to do to optimize the simulation result. Software used for simulation purpose is Autoform 7 Plus.

Steps to perform simulation

1. Import IGS files of the panel.
2. Set a plan of operation in sequence e.g. Drawing, Trimming, Restrike etc.
3. Set a tool definition such as punch, die and binder.
4. Set the pilot and blank.
5. Set the Parameters like sheet thickness, material, lubrication properties etc.
6. Kin check the simulation.
7. Start the Simulation.

Simulation's results check as per acceptability limit depending on the component area.

3.5 Results of Simulation after Drawing

Drawing is the operation where material is held by the binder then it is stretched over the punch profile so following are result of drawing operation.

3.5.1. Formability

The effect of formability of the sheet metal component shown in figure. Formability range, it shows colour definition to analysis the formability of panel as mentioned in validation results of formability. Here, the upper side of the panel needs some change feature activity to prove the panel because non-stretch on upper side tends to split the panel. Here, the green area shows the safe region which is acceptable. But the red area is split region which is not acceptable so need to modify the feature to remove red area. At the end, part area should show green area which will also increase the life of the press tool used. **(Refer Figure 14).**

3.5.2. Thinning

The effect of thinning of the sheet metal component shown in figure. Part result showed with colour variation of which limit is determined by part application area. Mostly for inner panel it is limit to 20% of sheet thickness. **(Refer Figure 15).**

3.5.3. Wrinkle

The effect of wrinkle of the sheet metal component shown in figure. Mostly for inner panel, it is limited to 3% of sheet thickness. **(Refer Figure 16).**

As per comparison, it cleared that the formability is good in EDD513 and the thinning is good in BSK 46. The effect of wrinkle in simulation result is good in all material. But the IFHS material shows average for formability and thinning. So, as per comparison IFHS material is better than other material used. It showed split on upper side of the panel. This condition is not valid. Here, part feature needs to change to get formability and thinning within acceptable limit.

3.6 Result of simulation after crashform operation

Crashform is the forming process, where no involvement of binder takes place. The formation takes by punch and die only in crashform operation. Here the panel got split in drawing operation so that panel need to change some geometric feature permanently or make some temporary feature in drawing and cleared in restrike operation to achieve result within acceptable limit. But there is also one way to achieve panel results without losing too much cost i.e. Crashform operation. The results after crashform operation are as shown in figure.**(Refer Figure 17).**

Here, the panel is in within acceptable range of limit. As the Front Roof Bow is an inner panel so it uses to support the roof and attach to BIW fixture so as per functionality of component its formability is ok and panel is proved.

4. CONCLUSION

In this paper, the validation of simulation software with actual result and optimization of parameter and process to prove the panel has introduced. This technology has advantages like optimization of material utilized by BIW designer for deciding panel split, reducing number of operations, reducing die cost, meeting target time, improve panel quality, press suitability study, reduce modification cost. The results of simulation have matched almost 90% with actual part data.

Then the optimization of process has introduced by using crashform operation instead of drawing process. The change in the process has reduced effort, cost and time by almost 30%. This change in process depends on the panel type whether it's inner or outer and its functionality. The comparison of the steel grade over simulation showed the result. Hence the material selection is important criteria and it depends upon forces act on the part as per its functionality.

The main disadvantages of the CAE tool are time and cost. In this process, lots of iteration needs to be done to achieve the results within acceptable limit. While proving the results of the panel, material flow is a key consideration. The material flow depends upon the simulation parameter. After optimizing all the parameters of the panel, results are achieved within the acceptable limits.

REFERENCES

[1] Miklós Tisza, Zsolt Lukács and Gaszton G. Gál (2008), Integrated process simulation and die-design in sheet metal forming, *International journal of Material forming*, v.1, pp 185-188.

[2] M. Tisza, (2007), Recent achievements in computer aided process planning and numerical modelling of sheet metal forming processes, *Journal of Achievement in material and manufacturing Engineering*, v.24, pp 435-442.

[3] Miklós Tisza and Zsolt Lukács (2013),Computer Aided Process Planning and Die Design in Simulation Environment in Sheet Metal Forming, *AIP conference proceeding* 1567, 100210.1063/1.4850140.

[4] Narendra S. Dhanjakar, Arun Namboodiri, Prasaana Muley,Gopal Musale (2011), Hyperworks Technology conference, HTC2011.

ISSN Online: 2394-0425; Volume 3 Issue 2 (2018)

LIST OF FIGURES:

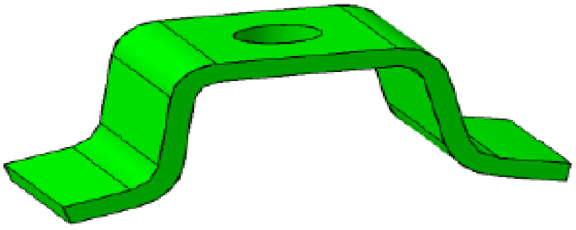

Figure 1. Bracket Heat Shield.

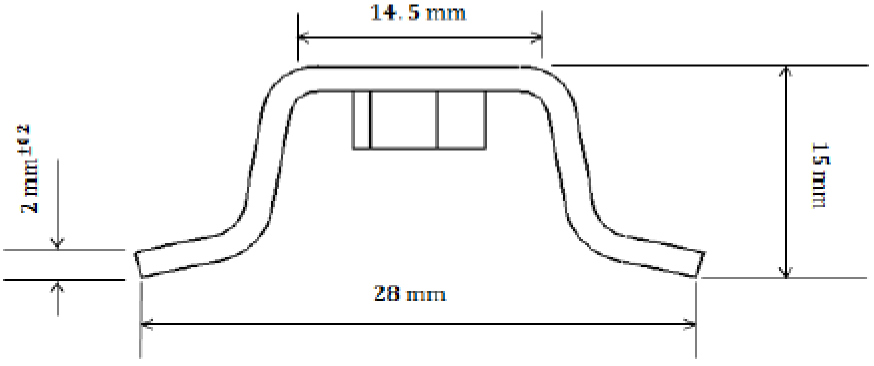

Figure 2. Part drawing for Bracket Heat Shield

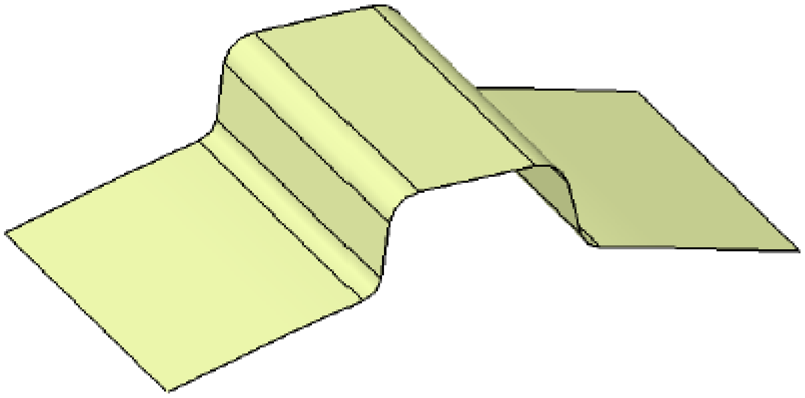

Figure 3. Generated Dieface

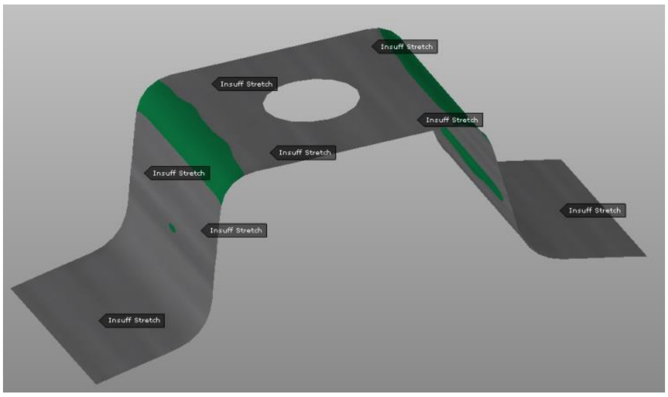

Figure 4. Result of Formability in Autoform.

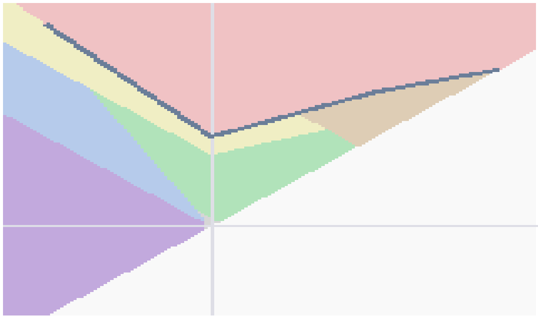

Figure 5. Formability limit Diagram.

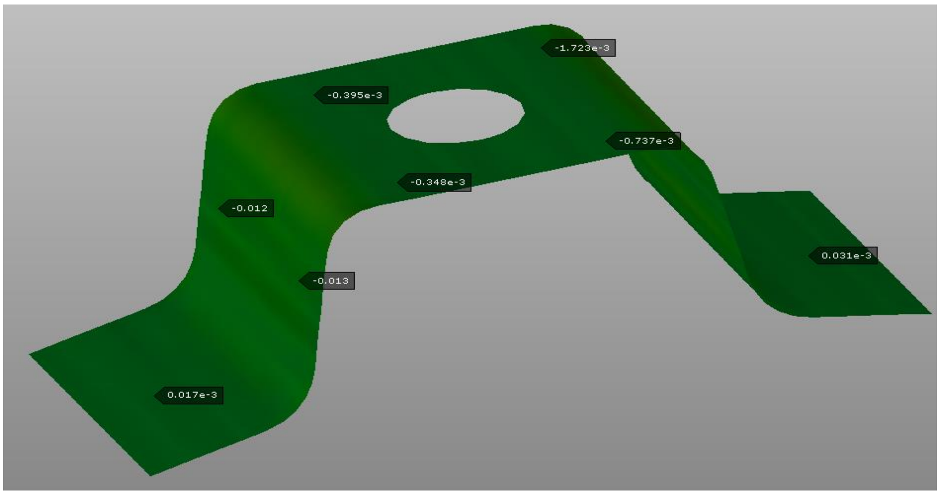

Figure 6. Result of thinning in Autoform

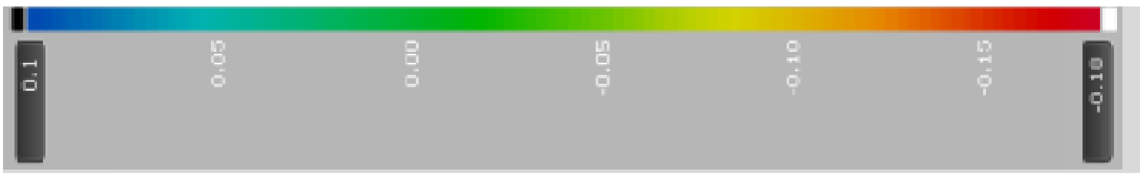

Figure 7. Thinning appearance limit scale.

ISSN Online: 2394-0425; Volume 3 Issue 2 (2018)

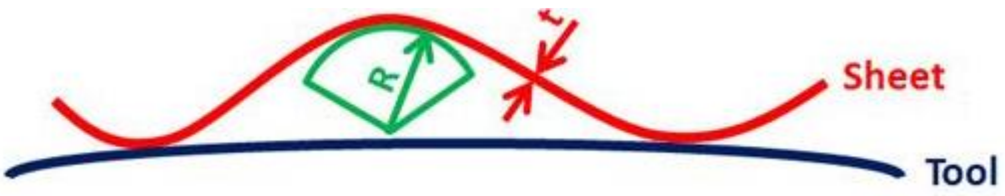

Figure 8. Formation of wrinkle.

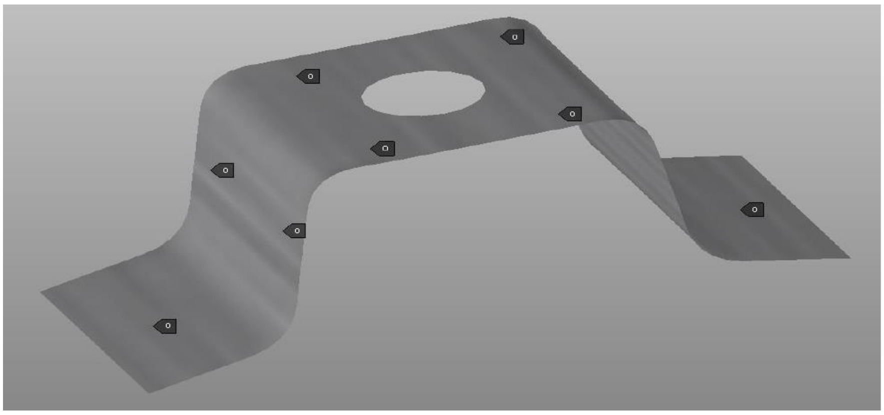

Figure 9. Result of wrinkle in Autoform

Points to be measured	Actual results (mm)	Simulation results (mm)
Point A	1.93	1.999
Point B	2.07	1.976
Point C	2.12	1.974
Point D	2.09	1.999
Point E	2.11	1.999
Point F	2.13	1.999
Point G	2.15	1.999
Point H	1.85	1.999

Figure 10. Comparison of actual result with simulations Result

ISSN Online: 2394-0425; Volume 3 Issue 2 (2018)

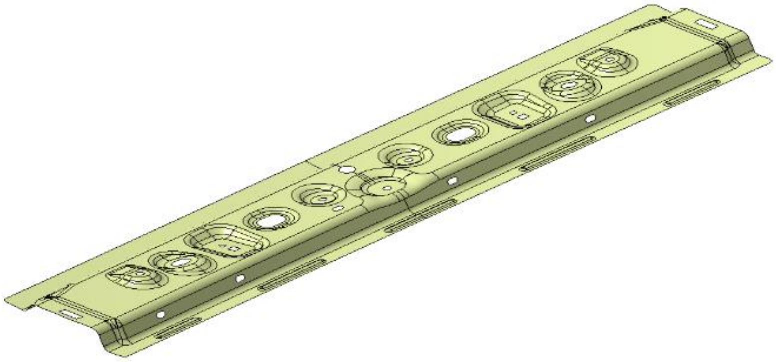

Figure 11. Front Roof Bow

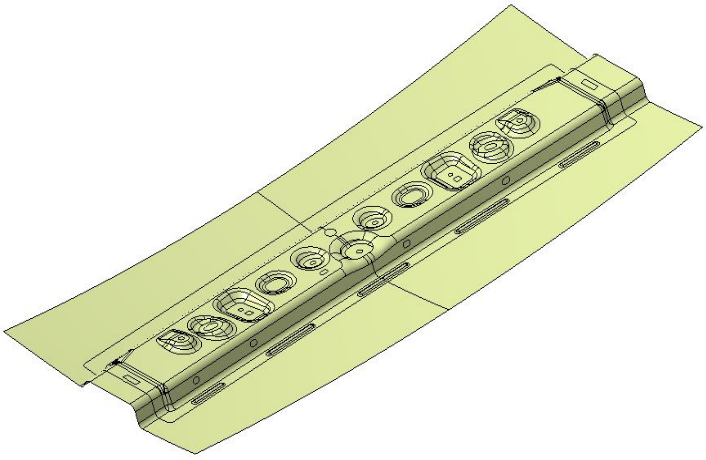

Figure 12. Developed dieface for Front Roof Bow

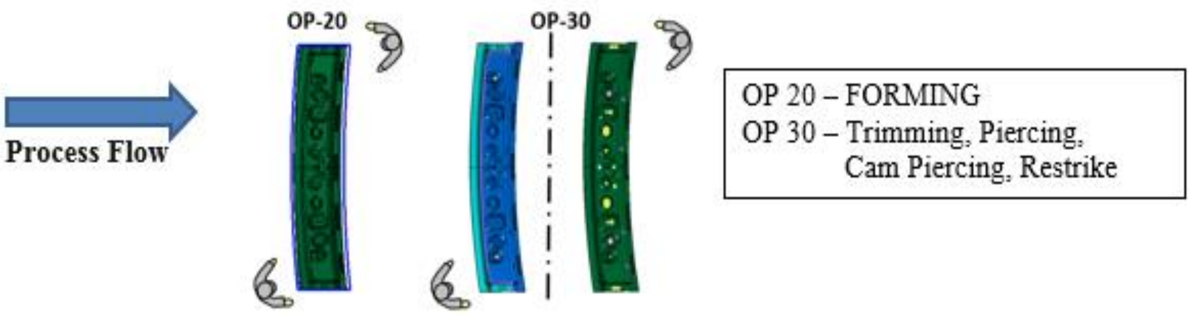

Figure 13. Process Plan

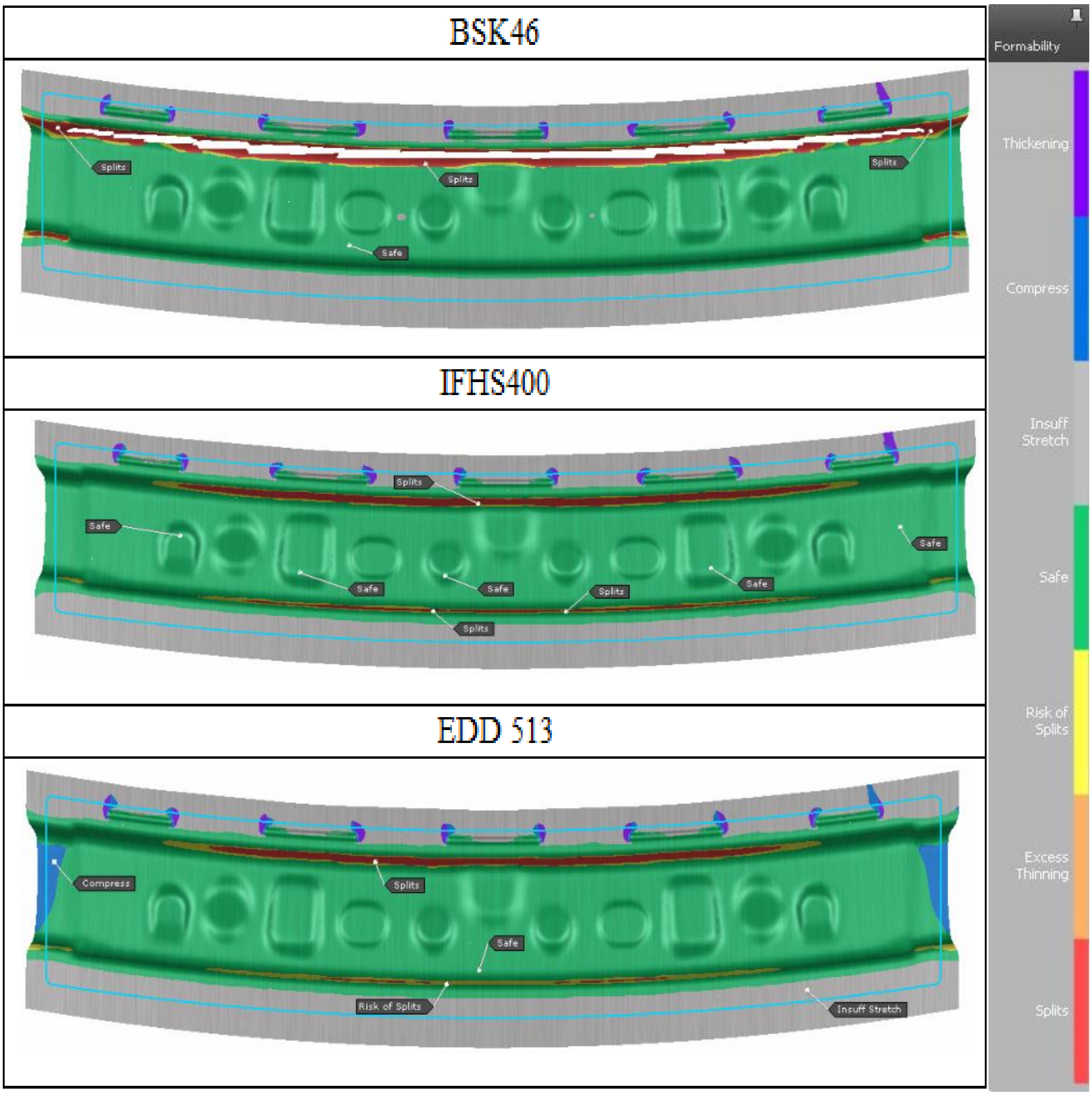

Figure 14. Effect on formability by different material

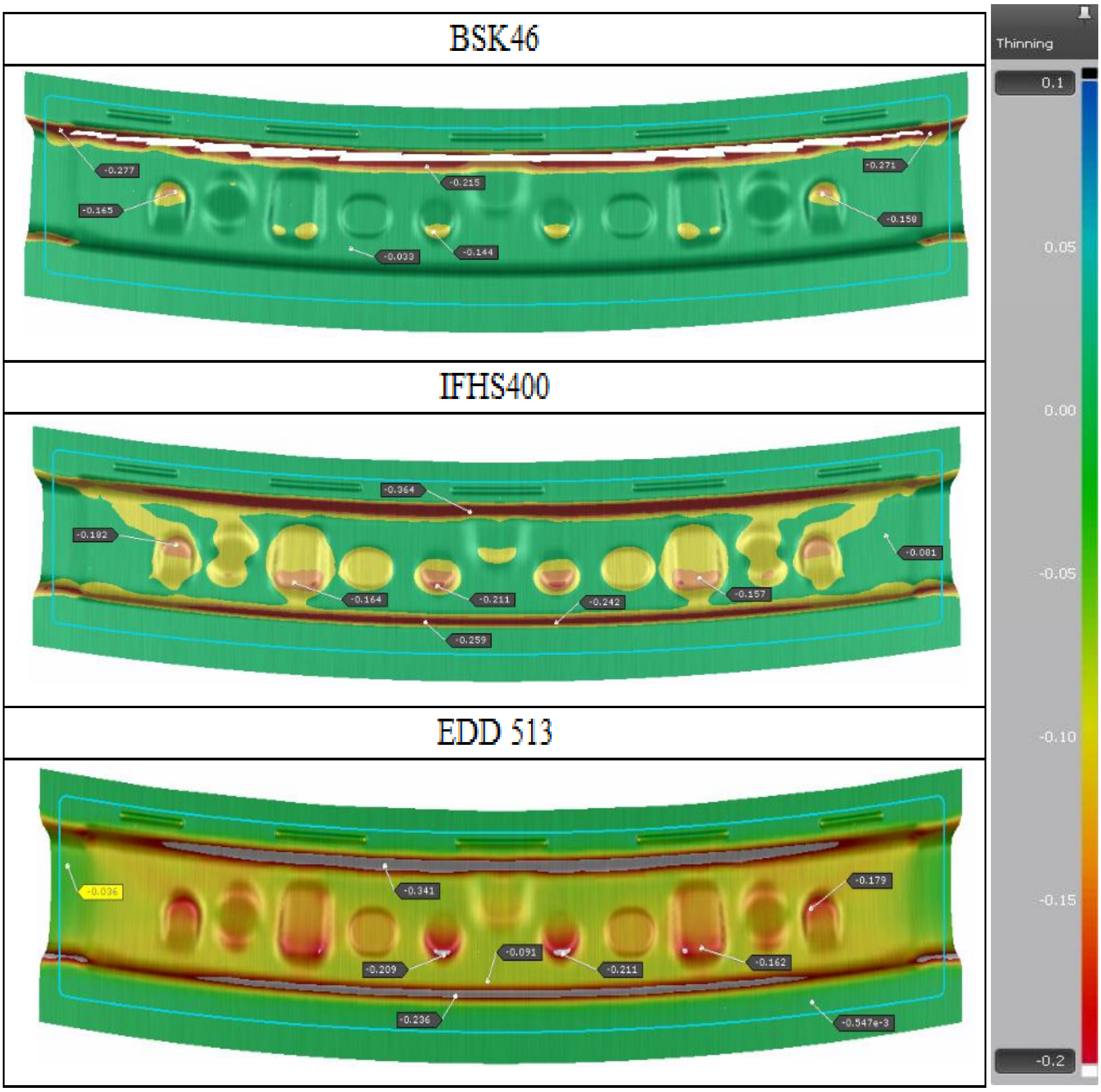

Figure 15. Effect on Thinning by different material

ISSN Online: 2394-0425; Volume 3 Issue 2 (2018)

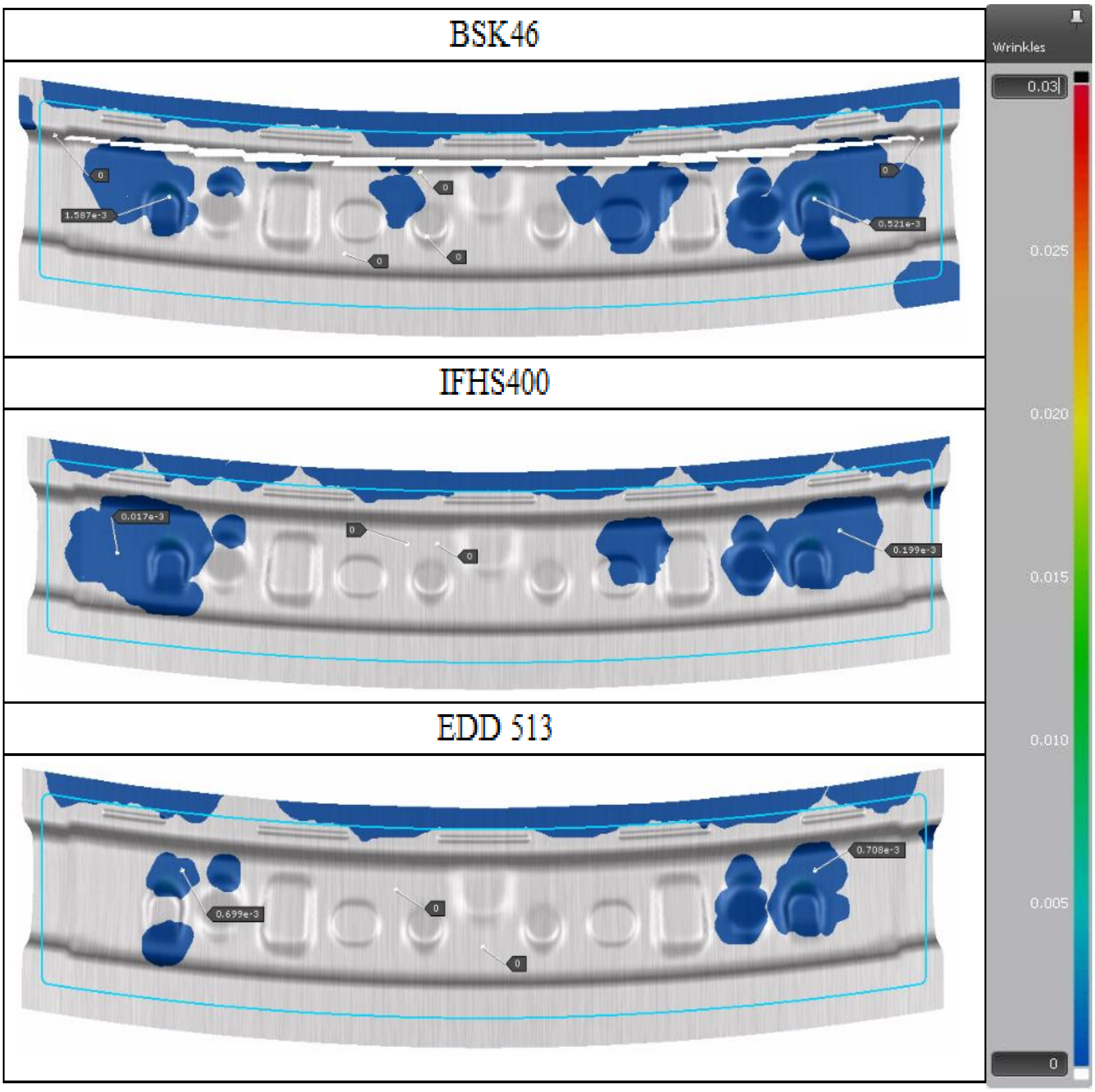

Figure 16. Effect on Wrinkle by different material

ISSN Online: 2394-0425; Volume 3 Issue 2 (2018)

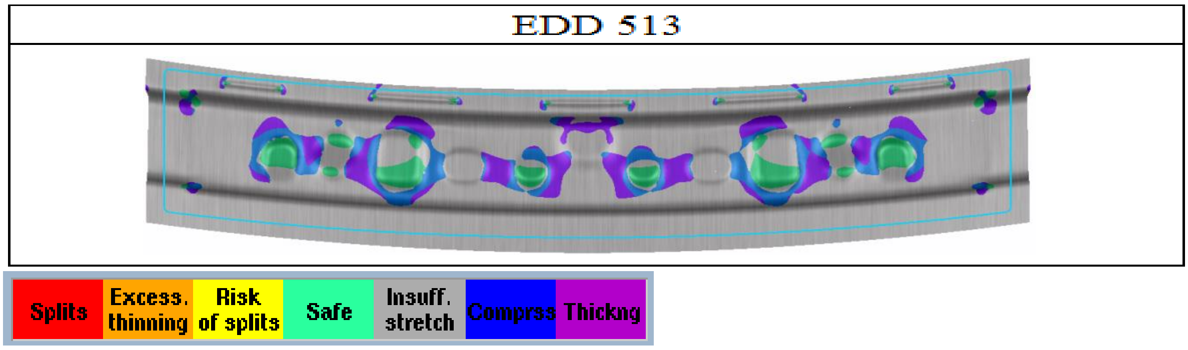

Figure 17. Formability for EDD513 by crashform

LIST OF TABLES:

Table 1. Properties of Bracket Heat Shield.

Material name: D513	
Properties	**Values**
n value	0.2
R value at 0/45/90 Degrees	1.29/1.33/1.3
Poisson's ratio	0.3
Young’s Modulus	2.1E+05MPa
Yield Strength	237MPa
UTS	334.2 MPa
K value (Strength coefficient)	580MPa

Table 2. Properties of Front Roof Bow

Material name: EDD513	
Properties	**Values**
n value(work Hardening Co-efficient)	0.241
R value at 0/45/90 Degrees	1.80/1.11/1.81
Poisson's ratio	0.3
Young’s Modulus(E)	210 GPa
Yield Strength	170 MPa

ISSN Online: 2394-0425; Volume 3 Issue 2 (2018)

UTS (Ultimate tensile strength)	270 MPa
K value (Strength coefficient)	0.5 GPa

Table 3. Effect on Formability, Thinning, Wrinkle for different material used on Front Roof Bow

Sr. No.	Material	Formability	Thinning	Wrinkle
1.	BSK46	• Part failure • Huge split on surface	• Thinning is in acceptable limit over radius in non-split area.	• Part splits • Rest area is in safe zone i.e. 3%of sheet thickness
2.	IFHS400	• split on the surface • good formability than BSK46	• Less thinning than EDD513 • Chances of split of radius.	• Part has Wrinkle in safe zone i.e. 3% of sheet thickness.
3.	EDD513	• Shows good formability over other material. • There is risk of split • Thickening also occurs	• Part has too much thinning out of acceptable limit. • Split occurs on the radius.	• Part has Wrinkle in safe zone i.e. 3% of sheet thickness.