Forming Simulation and Die Design in Sheet Metal Forming

Pratik Shah[1], D N Korade[2], K R Jagtap[3], P D Darade[4]

[1] *PG Student, Sinhgad Institute of Technology and Science, Pune, India*
[2,3,4]*Asst. Professor , Sinhgad Institute of Technology and Science, Pune, India*

---***---

Abstract:-*The design of new forming tools gets more problematic as the geometries get more complicated and the materials less formable. The idea with this project is to evaluate if an implementation of simulation software in the designing process, to simulate the forming process before actually developing the tools, could help to avoid expensive mistakes.*

In order to see the effect of die deformation on the forming of sheet metals, the draw-ins, strains, and spring backs of an automotive panel, following are simulated considering the die deformation. The die is designed considering the load exerted by the press.

The purpose of this paper is to give an overview about the recent achievements in sheet metal forming, and to avoid the spring-back by providing sufficient clearance, and to find optimum blank shape which in turn reduces the thinning of the sheet.

Key Words: **Die design, Sheet metal forming, Process simulation, Computer Aided Design, Forming Simulation.**

1. INTRODUCTION

In the recent years, the role and importance of metal forming processes in manufacturing industry have been continuously increasing primarily due to its material and cost effective nature. It is further emphasized by the recent advances in tools, materials and design, which in turn provide significant improvements in the mechanical properties and tolerances of the products. [1] Forming simulations are used throughout the entire process chain of sheet metal forming. The simulation allows a part designer to estimate the formability of a sheet metal part already during the design phase, which results in the design of a part which is easy to produce. It is also possible to see how the process parameters must be adjusted in order to guarantee optimal drawing results. M Tisza and T Altan [1, 2] stated that the application of various Computer Aided Engineering [CAE] techniques practically covers the full product development cycle from the conceptual product design through the process planning and die design up to the manufacturing phase of the production. CAE techniques are widely used in sheet metal forming, for example to predict the formability, to determine the type and sequences of manufacturing processes and their parameters, to design forming tools, etc.

The importance of the application of CAE tools becoming more and more important as the manufactured parts are becoming ever increasingly complex. As the need for the widespread application of CAE techniques driven by the demand of global competitiveness accelerates, the need for a robust and streamlined process and die design engineering (PDDE) becomes more and more crucial.

Y.T.Keum [3] carried out the forming and the spring-back analyses of an automotive fender panel by coupling the forming analysis and the structural analysis for considering die deformation. A Alf and Andersson [4, 5] carried out an experiment to find out comparison of sheet metal forming simulation and try-out tools in design of forming tools. And as a result he said that the use of sheet-metal-forming simulation leads to a significant reduction in both cost and time compared with the use of try-out tools.

Sheet metal parts are light weight and can have versatile shape due to its low cost and generally good strength and formability characteristics. Many sheet metal forming processes are used to produce parts and shapes and there is usually more than one method of manufacturing a sheet metal from a given material. Various sheet metal forming process are shearing process, forming process and finishing process.

The main problem in die deign are the design of die and punch due to complex shape, weight reduction is main challenge and spring-back is a particularly critical aspect of sheet metal forming. Even relatively small amounts of spring-back in structures that are formed to a significant depth may cause the blank to distort to the point that tolerances cannot be held.

The main objective of this project is to simulate the different stages of forming process, achieve weight reduction, increase production rate and to find good blank shape.

2. PROCESS PLANNING AND DIE DESIGN IN SHEET METAL FORMING

One of the main drawbacks in industrial practice hindering the even more wide application of simulation techniques that the output results of simulation packages are not usually directly and easily usable for computer aided die design. Obviously, there are tremendous efforts to successfully link CAD and FEM systems. This solution requires a fully integrated approach of computer aided product design, process planning and die design, as well as the finite element simulation of the forming processes. It means that simulation tools should be efficiently used throughout the whole product development cycle [6].

This concept will be illustrated through the examples of automotive part production. In our practice, we use CATIA as a CAD system for supporting the Process Planning and Die Design tasks and the Auto Form is used as the numerical simulation tool, however, the principles applied here can be similarly adopted by using different CAD and simulation packages, too. Before analysing this integrated solution, let's summarize the main features of forming process planning and die design in so-called conventional CAD environment.

2.1 Process Planning and Die Design in Conventional CAD Environment

Stamping industry applies CAD techniques both in the process planning and die design already for many years. However, in a "traditional" CAD environment, these are practically stand-alone solutions, i.e. for example a knowledge based process planning solution is applied for the determination of the necessary types of forming processes, even in some cases, the forming sequences can be determined in this way together with the appropriate process parameters. Therefore, before it goes to the production line, usually a time and cost consuming try-out phase follows, as it is shown in Figure 1.

If the try-out is successful, i.e. the die produces parts with no stamping defects, it will be sent to the stamping plant for production. On the other hand, if splitting or wrinkling occur during the tryout, the die set needs to be reworked. It means that we have to return first to rework the die construction by changing the critical die parameters (e.g. die radii, drawing gap, etc.). If it does not solve the problem, a new die design, or a new process planning is required. Some cases, we have to go back even to the product design stage to modify the product parameters.

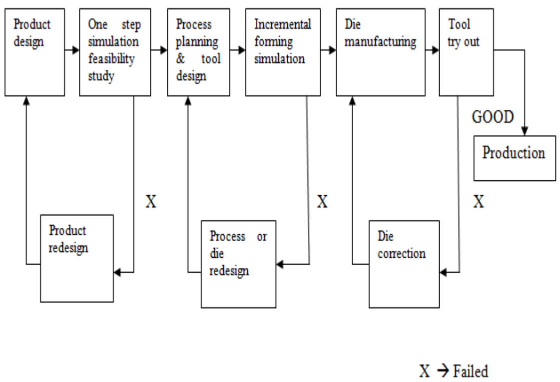

Figure 1 : Flow chart of process planning and die design.

2.2 Simulation based process planning and die design

Due to the global competition and this is particularly valid for the automotive industry there is an overall demand to improve the efficiency in both the process planning and in the die design phase, as well as to reduce the time and product development costs and to shorten the lead times. It requires the efficient use of simulation techniques from the earliest stage of product development, to give feedback from each step to make the necessary corrections and improvement when it takes the least cost [7]. This principle is illustrated in the schematic flow chart of simulation based process planning and die design as shown in Figure 2.

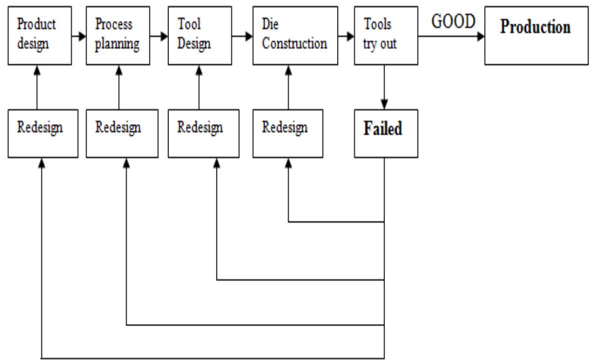

Figure 2 : Work flow in simulation based process planning and die design.

With this approach, stamping defects may be minimized and even eliminated before the real die construction stage. If any correction or redesign is needed, it can be done immediately, with a very short feedback time, thus it leads to a much smoother die try-out if necessary at all and to significantly shorter lead times with less development costs. However, even with this approach, there are some further shortfalls in the die design process, since most of the simulation programs do not provide die construction in sufficient details, which can be easily used in most of the CAD systems to complete the die design task. This shortage may be overcome by integrating the CAD and FEM systems through a special interface module which can provide a smooth, continuous and reliable data exchange between the two important parts of design process.

2.3 Die design

A piercing die has to be design for the following Reinforcement Wheel Arch CAD model. Each component has 3 holes of different diameter. The following reinforcement wheel arch [component 1] and [component 2] are shown in Figure 3 and Figure 4 respectively.

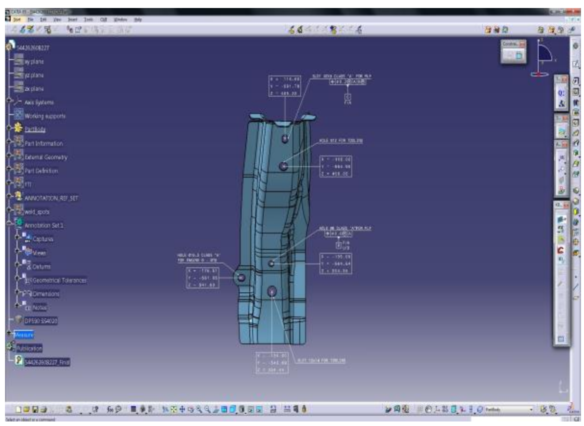

Figure 3 : CAD model of component 1

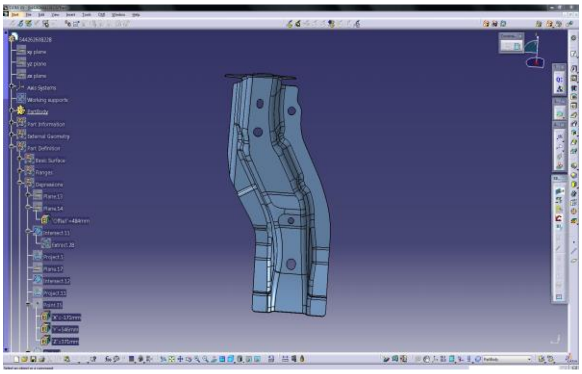

Figure 4 : CAD model of component 2

The following component 1 and component 2 is a “reinforcement wheel arch” which comprises of two holes and one slot of ‘class A’ grade.

Table 1 : Hole Dimension

	Component 1	Component 2
Hole A	8 mm	8 mm
Hole B	10.5 mm	12.5 mm
Hole C	12 x 8 mm	12 x 8 mm

A piercing die comprises of two main parts, i.e. Lower Die & Upper Die, shown in Figure 5 and Figure 6 respectively.

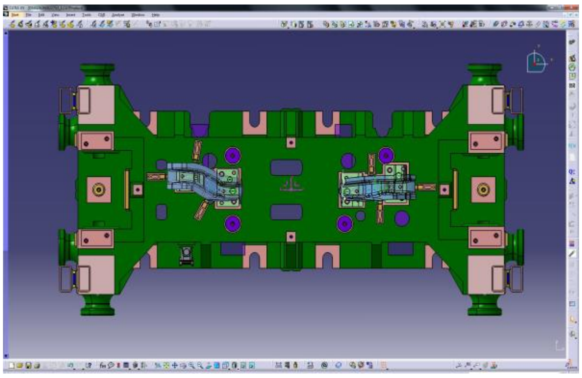

Figure 5: CAD model of Lower Die.

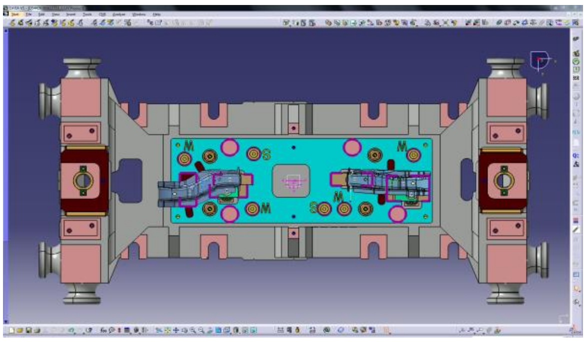

Figure 6 : CAD model of Upper Die.

Figure 3 and Figure 4 shows “Reinforcement Wheel Arch” component for which cutting load and stripping load are calculated.

- Die material: FG 260
- Component material: DP590
- Sheet thickness = 1.2 mm

■ Cutting load [F] = L * t * τ_b

Where, F = cutting load

L = cutting parameter [total parameter of 3 holes]

t = sheet thickness [1.2mm]

τ_b = sheet shear strength

F = 220.780*1.2*590

F = 156312.24 N

F = 15631 Kg

■ Stripping load = 10% of cutting load

15631/10 = 1563.1 Kg = 1.5 tonne

■ Clearance = 0.095 mm (from table)

■ Entrance = 6mm

■ Stripper stroke = 10mm + Sheet thickness + entrance

= 10mm + 1.2mm + 6mm = 17.2mm ≈ 15mm

➢ Coil spring is used to balance the exerting force.

Selection of coil spring is done by using the MiSUMi standard part handbook.

SWM 30.50 coil spring is used (from handbook) which exert the force of 168.75 Kgf, shown in Figure 7 below

■ Stripper force = stripping load / spring force = 1563/168.75 = 9.27 ≈ 10

Therefore, 10 springs are used to balance the force.

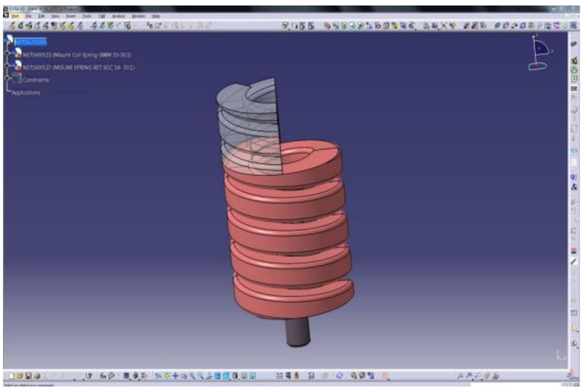

Figure 7: SWM 30.50 Coil Spring

- Weight calculation :

- For old die :-

Total weight = Die Volume * weight factor * specific gravity of steel

Die volume = 1700mm*600mm*380mm = $387.6*10^6 mm^3$

Weight factor = 0.4

Specific gravity of steel = 8 gms/cm^3

Total weight = $(387.6*10^6)*0.4*(8*10^{-6})$ =1240.32 Kg ≈ 1.1 tonne [from CATIA]

Combine weight = 2.2 tonne [for single die]

Combine weight = 4.5 tonne [for two die]

- For proposed die :-

Total weight = Die Volume * weight factor * specific gravity of steel

Die volume = 1950mm*800mm*380mm = $592.8*10^6 mm^3$

Weight factor = 0.4

Specific gravity of steel = 8 gms/cm^3

Total weight = $(592.8*10^6)*0.4*(8*10^{-6})$ =1896.96 Kg ≈ 1.5 tonne [from CATIA]

Combine weight of die = 3 Tonne [for single die]

- Difference between the old die and proposed die

Combine weight of the old die = 4.5 tonne

Combine weight of the new die = 3 tonne

Weight difference = 1.5 tonne

% weight reduction = $\frac{\text{old die weight} - \text{new die weight}}{\text{old die weight}}$ X 100

$= \frac{4.5-3}{4.5}$ X 100 = 33.33 %

- Production Rate

- Old Die [component 1]
Production rate = 4 component/min
Therefore, for 500 component 125 minute

- Old Die [component 2]
Production rate = 3 component/min
Therefore, for 500 component 166 minute
All together 291 minute for 1000 component

- Proposed Die Production rate = 6 component/min

Therefore, for 1000 component 166 minute

% Increase Production Rate = $\frac{291-166}{291}$ x 100 = 42.95 % ≈ 43

3. FORMING SIMULATION

Firstly, the component is imported into simulation software (Auto Form) and then the optimum blank shape is derived. Then the forming material is assigned from the material library.

- Component 1 [Reinforcement Wheel Arch]

Firstly, the simulation for component is done to find optimum blank shape. Secondly, the simulation for crash forms at various stroke of press. Thirdly for thickness plot and finally for the spring-back plot.

Table 2: Material Properties [DP 590]

n value	0.16
R value at 0/45/90° & R-Bar Value	0.82
Poisson's Ratio	0.3
Young's Modulus	200 Gpa
Yield Strength	0.41 Gpa
UTS	>0.5
K Value (strength coefficient)	1.25 Gpa

Table 3: Simulation Set-up (component 1)

Material	DP590
Thickness	1.2 mm
Master surface	Upper
Blank size	350 x 165 x 1.2
Material yield	71.88%
Coefficient of friction	0.14

Table 2 and Table 3 give the information about the selected sheet metal material and simulation setup specification for component 1 respectively.

Optimum blank shape has many evident advantages. The optimum blank not only improves formability and product quality but also reduces material cost, number of trials in the try-out stage and product development period. Moreover, the optimum blank shape leads to the prevention of tearing, the uniform thickness distribution and the reduction of the press load during drawing. It is shown in Figure 8 below.

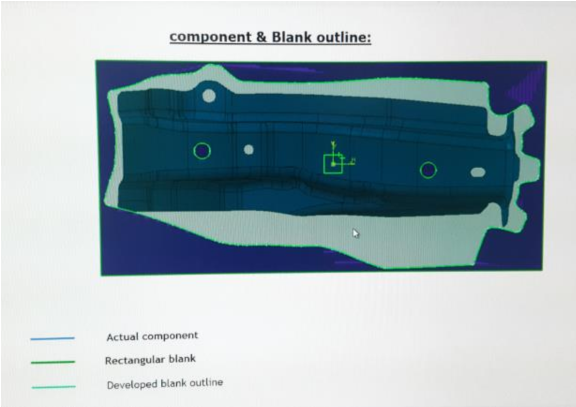

Figure 8 : Component & Blank Outline

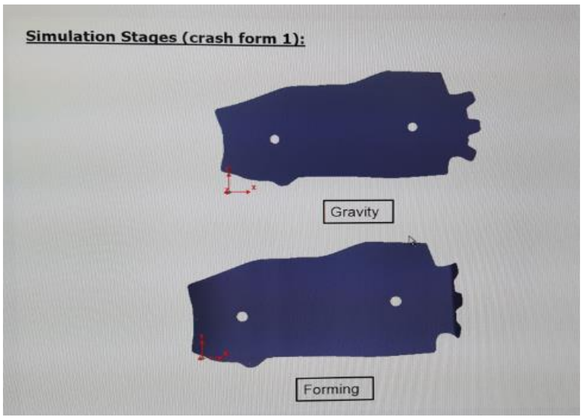

Figure 9: Crash forming simulation stages (crash form 1)

In the Figure 9, Figure 10 and Figure 11 Crash forming simulation at gravity state and at forming state is shown. A crash form means that a male punch and a female die are closed together without any springs, cushions. At gravity state the blank is placed at the surface of lower die having surface contact with the upper die, where press does not exert any force on blank, whereas at forming state the press exerts the force without lowering the upper die. Parts can be "crashed" from a square blank and then trimmed OR a blank can be trimmed first and then crashed. The latter is the cheapest tooling method. Further at closing and -80mm state the press tends to lower the upper die causing the forming operation. Again at -70mm, -60mm and -30 mm the press exert the force on the blank which continues the forming process.

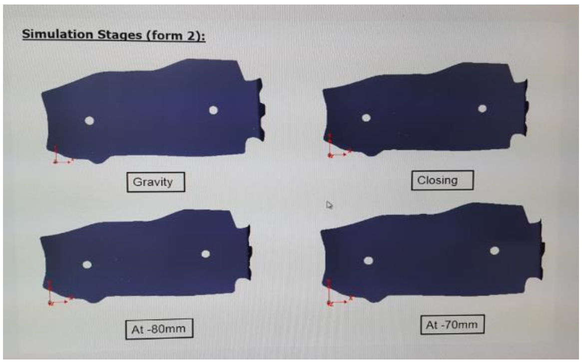

Figure 10: Forming simulation Stages (form 2).

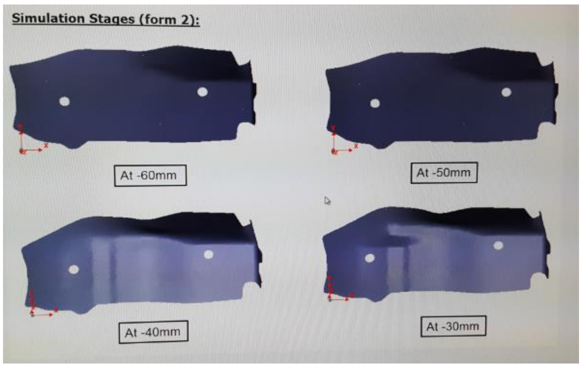

Figure 11 : Forming simulation Stages (form 2).

The lowering of the press stop when the punch reach at the BDC and the final component is formed with some minor cracks which are negligible shown in Figure 12 and Figure 13 respectively.

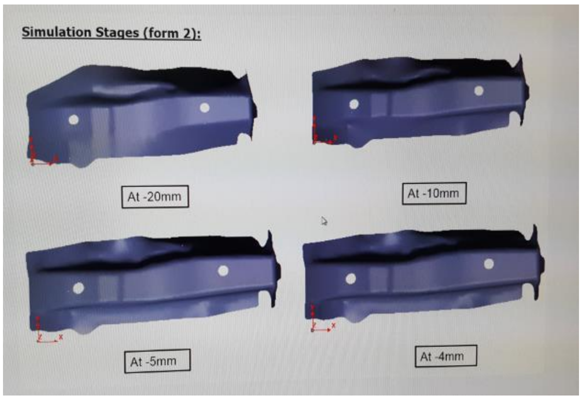

Figure 12: Forming simulation Stages (form 2).

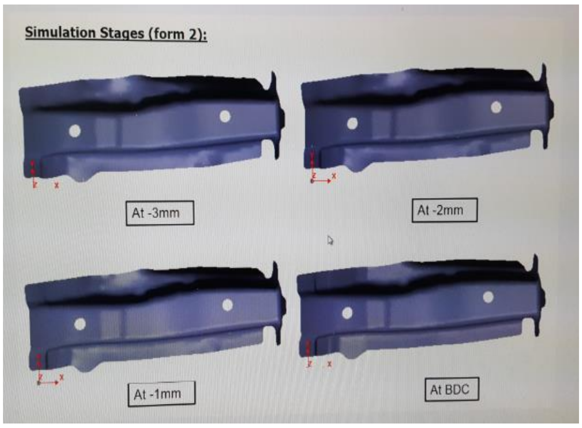

Figure 13: Forming simulation Stages (form 2).

Further, simulation is done for thickness plot and spring-back plot.

In Figure 14 Thickness Plot is shown, in which minor cracks are shown at the right most corner of the component. The thinning is greater than equal to 0.2mm which is negligible, which can be overcome by reliving upper steel and finding optimum blank shape.

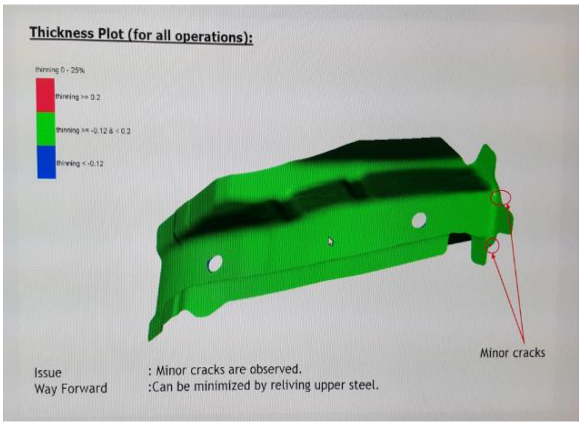

Figure 14: Thickness Plot

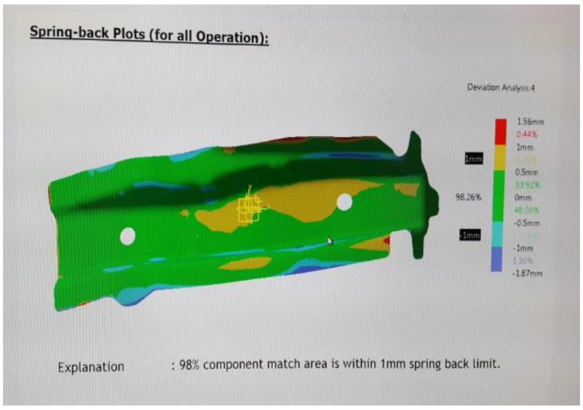

Figure 15: Spring-back Plot

Spring-back is a major issue when it comes to metal forming and production. Advanced high-strength steels bring additional formability concerns, so knowing how to predict the magnitude and direction of spring-back through accurate simulation is more important than ever before. This can reduce production cost by lowering the amount of time invested and re-cuts necessary.

In Figure 15 spring-back plot is shown which state that 98% of component match area is within limit.

Spring-back compensation is the design and adjustment of the tooling faces to allow for the expected spring-back, usually "over forming" the part such that the shape of the part is correct after spring-back occurs. For simple forming operations (such as bending) compensation seems straightforward, and the old trial and error method may yield good results quickly. However, for today's complex 3D sheet metal shapes, anything other than a straight line bend requires advanced simulation technology to predict and compensate for spring-back.

AutoForm uses a sophisticated iterative method to apply spring-back compensation more than once, applying, adjusting, simulating and re-calculating spring-back many times over until a specific result is achieved.

- ❖ Component 2 [Reinforcement Wheel Arch]

Table 4: Material properties [DP 590]

n value	0.16
R value at 0/45/90° & R-Bar Value	0.82
Poisson's Ratio	0.3
Young's Modulus	200 Gpa
Yield Strength	0.41 Gpa
UTS	>0.5
K Value (strength coefficient)	1.25 Gpa

Firstly, the simulation for component is to be done to find optimum blank shape. Secondly, the simulation for crash forms at various stroke of press. Thirdly, for thickness plot, and finally for the spring-back plot. Table 4 and Table 5 give the information about the selected sheet metal material and simulation setup specification for component 2 respectively.

Table 5: Simulation Set-up (component 2)

Material	DP590
Thickness	1.2 mm
Master surface	Lower
Blank size	350 x 215 x 1.2
Material yield	62.42%
Coefficient of friction	0.14
Binder stroke for Form	80 mm
Blank holding force for Form	25 T

Optimum blank shape has many evident advantages. The optimum blank not only improves formability and product quality but also reduces material cost, number of trials in the try-out stage and product development period. Moreover, the optimum blank shape leads to the prevention of tearing, the uniform thickness distribution and the reduction of the press load during drawing. It is shown in Figure 16 below.

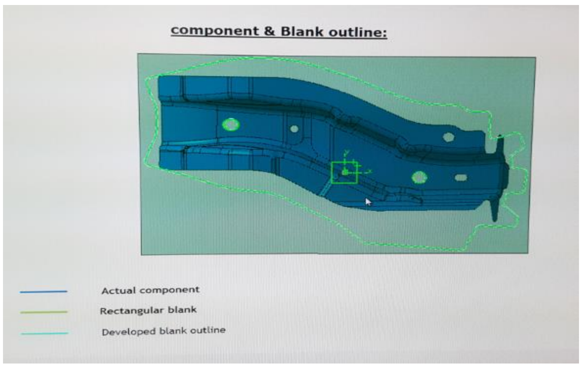

Figure 16: Component & blank outline.

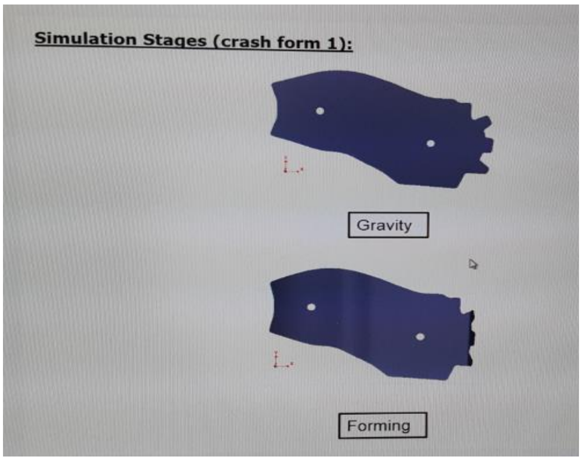

Figure 17: Crash forming simulation (crash form 1).

In the Figure 17, Figure 18 and Figure 19 Crash forming simulation at gravity state and at forming state is shown.

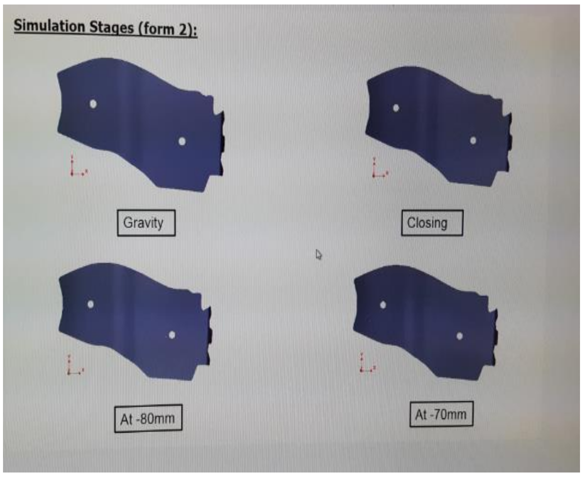

Figure 18: Forming simulation stages (form 2).

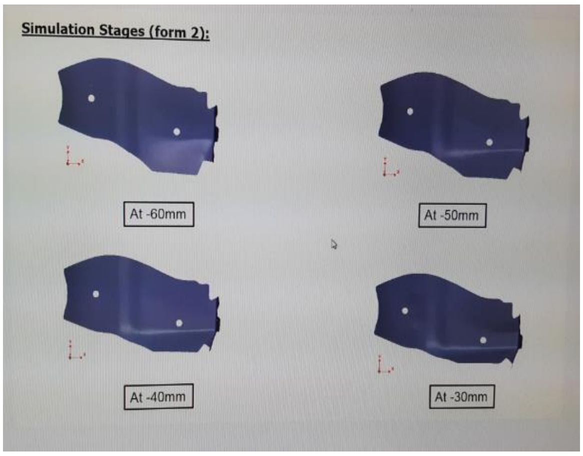

Figure 19: Forming simulation stages (form 2)

Further at closing and -80mm state the press tends to lower the upper die causing the forming operation. Again at -70mm, -60mm and -30 mm the press exert the force on the blank which continues the forming process.

The lowering of the press stop when the punch reach at the BDC and the final component is formed with some minor cracks which are negligible shown in Figure 20 and Figure 21 respectively.

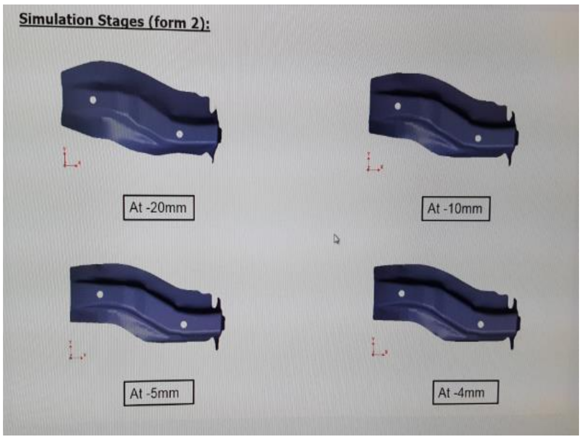

Figure 20: Forming simulation stages (form 2).

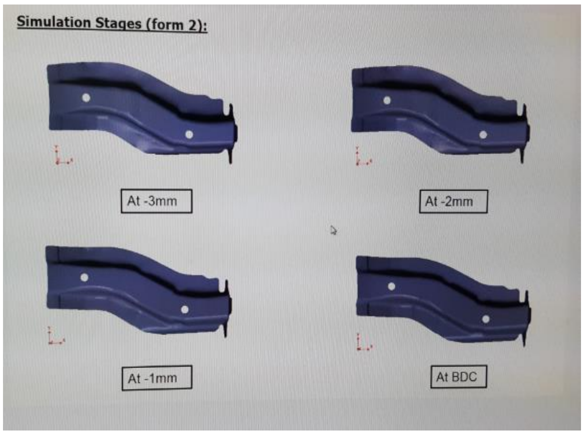

Figure 21: Forming simulation stages (form 2).

Further, simulation is done for thickness plot and spring-back plot.

In Figure 22 Thickness Plot is shown, in which minor cracks are shown at the right most corner of the component. The thinning is greater than equal to 0.2mm which is negligible, which can be overcome by reliving upper steel and finding optimum blank shape.

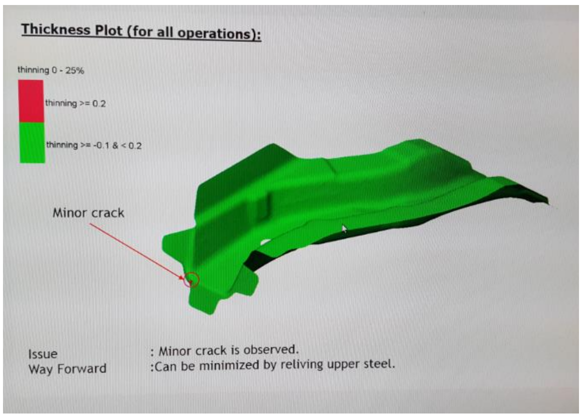

Figure 22: Thickness plot.

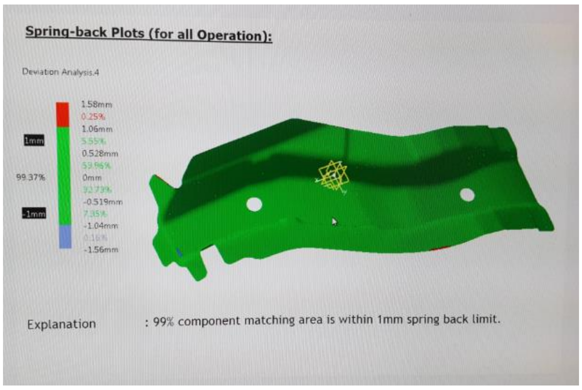

Figure 23: Spring back plot.

Spring-back is a major issue when it comes to metal forming and production. Advanced high-strength steels bring additional formability concerns, so knowing how to predict the magnitude and direction of spring-back through accurate simulation is more important than ever before. This can reduce production cost by lowering the amount of time invested and re-cuts necessary.

In Figure 23 spring-back plot is shown which state that 98% of component match area is within limit.

4. RESULT AND DISCUSSION

As per the calculation it is derived that the overall weight of the die is reduced by 1.5 tonne i.e. 33.33 % of the total weight of the old die. By removing the excess material from the casting though maintaining the strength of the die weight reduction is achieved. Thus 33.33% of weight reduction is achieved.

Production rate is also increased by combining the two small component die into a single large die. Production rate is increased by 43 %.

It is seen that 98% of component match area is within 1mm limit for component 1 and 99% for component 2 [from Figure 15 and Figure 23]

From figure 14 and figure 22 it is seen that minor cracks are observed which are considered as negligible.

As it is observed that the spring-back and thinning of sheet metal while forming process going on is under control, therefore the proposed die is safe to use.

And it is also seen that the proposed is lighter than the existing old die, which result in cost reduction.

5. CONCLUSIONS

The purpose of this project is to give a general overview about the recent achievements in the field of sheet metal forming and to introduce some special results in this development activity. Therefore, in this project, an integrated process simulation and die design system was analyzed. The proposed integrated solutions have great practical importance to improve the global competitiveness of sheet metal forming in the very important segment of industry. The concept described in this project may have specific value both for process planning and die design engineers.

After doing this research work following conclusion can be drawn

1. Overall weight of the die is reduced by 1.5 Tonne (i.e. by 33.33 %).
2. Cost reduction is achieved by reducing the total weight.
3. Production rate is increased by combining two components in a single die by 43 %.
4. Minor cracks are found which are negligible.
5. It is seen that 98 % component match area is within 1mm spring-back limit for component 1 and 99 % component match area is within 1mm spring-back limit for component 2, which is acceptable.

REFERENCES

1. M Tisza, Z Lukács and G G. Gál, International Deep-drawing Research Group IDDRG 2007 International Conference 21-23 May 2007.pp 235-240.
2. T. Altan, V. Vasquez: New Concepts in Die Design – Physical and Computer Modelling, Journal of Material Processing. Technology, Columbus, V. 98. (2000) pp. 212-223.
3. Y.T.Keum, I.H.Ahn, I.K.Lee, M.H.Song, S.O.Kwon, And J.S.Park, © 2005 American institute of Physics, CP778 Volume A, Numisheet 2005.pp 90-95.
4. A Alf, Comparison Of Sheet Metal Forming Simulation And Try-Out Tools In Design Of Forming Tools, Journal of Engineering Design, Sweden, V. 15. (2004) pp. 551-561.
5. Andersson, Information Exchange Within Tool Design And Sheet Metal Forming, J. Of Engineering Design, 2001. V. 12. No. 4. pp. 283-291.
6. M. Tisza: Numerical Modeling And Simulation: Academic and Industrial Perspectives, Materials Science Forum, V. 473-474. (2005) pp. 407-414.
7. J. J. Yin, Y. S. Shin, H.Y. Kim, “Analysis On The Bending Deflection Of The Blank Holder In Automotive Body Panel Draw Die”, Transactions Of The Korean Society Of Machine Tool Engineers Vol.10, No.3, 2001, pp68-74.