Finite Element Modeling for Numerical Simulation of Multi Step Forming of Wheel Disc and Control of Excessive Thinning

Prashantkumar S.Hiremath[1,a], Shridhar Kurse[2,a], Laxminarayana H.V.[3,a],Vasantha kumar M[4,b], Ravindra J L[5,b].
[1] Post Graduate Student,[2] Associate. Professor,[3] Professor,[4,5]Senior Project Manager.
[a] Dept. of Mechanical Engineering, Dayananda Sagar College of Engineering, Bangalore
[b] Dept. of Product Design and Development, Altair India Bangalore

***Abstract*—The aim of this paper is numerical simulation of multi step sheet metal forming process of a wheel disc and an effort made to propose an efficient method to optimize the sheet metal stamping process to obtain improved quality of a product. The main topic of the work is the prevention of excessive part thinning and the control of springback phenomena; thus, thinning and springback are the objective functions taken into account. The blank holder force value was considered as process design variable. The approach proposed in this work is a multi-objective optimization problem consisting of integration among Finite Element Numerical Simulation and Response Surface Methodology.**

Keywords—* *Wheel Disc Forming; Excessive Thinning; HyperForm; Radioss;

I. INTRODUCTION

Metal Stamping is a forming process by plastic deformation of a metal surface carried by punch in a die. The surface is transferred by molecular displacement of matter. To avoid trial and error tryout procedures, finite element simulation method has been used in sheet metal stamping in a wide range to evaluate the deformation defects and optimize the design.

Nowadays, in a complex 3D industrial forming processes design, the total compensation (or at least reduction) of the springback distortions and excessive thinning is a crucial Sheet metal stamping simulation is necessary in order to predict the part failure, understanding the % thinning, optimized blank development, press tonnage calculation and to reduce product development cost, product development time and also the cost of Rework. Minimization of excessive thinning and springback distortions leads to minimize the costs. The actual application of any optimization technique should take into account the possibility to properly deal with all the conflicting goals. Some approaches have been aimed to determine a cluster of possible optimal solutions by applying multi-objective techniques in sheet metal stamping optimization.

II. REVIEW OF RELATED RESEARCH

Many research studies have been carried out to analyze the sheet metal forming simulation and to optimize the forming parameters, geometrical shapes and material parameters to improve the forming quality in the stamping process based on the simulation results of finite element modeling (FEM).

Y. Huangan et al.[1]outlined Minimization of the thickness variation in multi-step sheet metal stamping. The aim of this paper was to develop efficient method to optimize the intermediate tool surfaces in the multi-step sheet metal stamping process to obtain improved quality of a product at the end of forming. The proposed method is based on the combination of finite element modeling (FEM) and the response surface method (RSM).

A multi-objective stochastic optimization approach was presented by L. Marrettan et al.[2]. The aim of this paper was to develop a design tool for stamping processes, which is able to deal with the scattering of the final part quality due to the inner variability of such operations.

Two-step method of forming complex shapes from sheet metal was presented by Sergey F. Golovashchenko et al.[3]. A two-step method of forming a part and a method of designing a preformed shape are being discussed.

T. Cwiekala et al.[4] outlined accurate deep drawing simulation by combining analytical approaches. The aim of this paper was to development of an analytical simulation method for deep drawing processes.

Fast thickness prediction and blank design in sheet metal forming was presented by B.T. Tang et al.[5]. In this paper, a robust energy-based 3D mesh mapping algorithm is used to obtain the initial solution and is followed by a reverse deformation method to improve its accuracy. The novel initial solution scheme can consider the material and the process parameters, and thus lead to fewer Newton–Raphson iterations.

III. PROBLEM STATEMENT

The accompanying sketch shown in the Figure1 is a Wheel Disc of a car, it is intended to manufacture this component using multi step forming process. Modeling and simulation of the process to arrive at acceptable component is essential. The main aim of the paper is to simulate the sheet metal forming stages of wheel disc and optimization of necessary forming step.

The main focus is on the prevention of excessive part thinning and the control of springback phenomena; thus, thinning and springback are the objective functions taken in to consideration. The blank holder force (BHF) value was considered as process design variable. Geometric details and material properties have shown in the Fig 2 and Table 1 respectively and thickness of the wheel disc is 3.5mm.

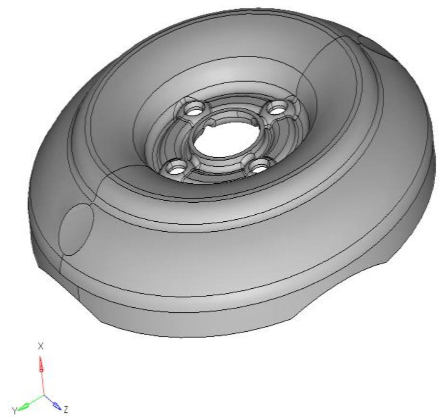


Fig 1: Wheel disc model: Isometric view

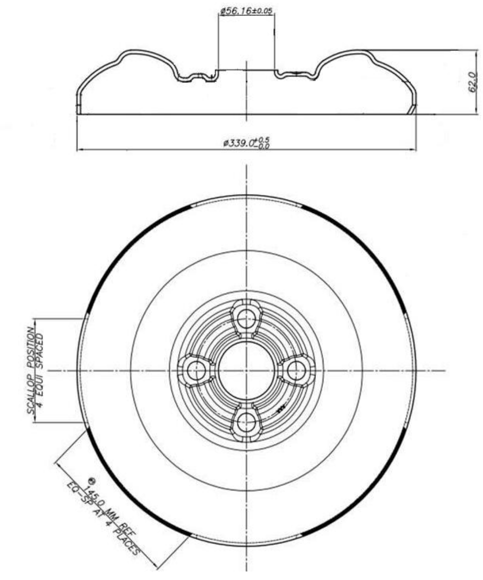


Fig 2: Geometric details of wheel disc

Table 1: Mechanical properties of wheel disc material

Yield stress, σ_y (MPa)	521
Ultimate Tensile Strength (MPa)	610
Plastic strain anisotropic factor ‘r’	0.8
Strain hardening exponent ‘n’	0.1
Strain hardening co-efficient ‘K’(MPa)	824
Coefficient of friction,μ	0.125
Modulus of elasticity, E (MPa)	2.07e5
Density, ρ (g/cm3)	7.8
Poisson’s ratio, v	0.28
True ultimate tensile strength (MPa)	701.5

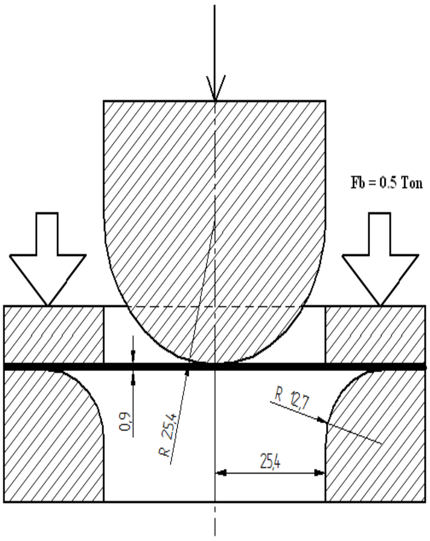
IV. AIM AND OBJECTIVE

The aim of the work is to perform multi step incremental forming simulation of a wheel disc using HyperForm and to propose an optimization approach in order to control the excessive part thinning and springback phenomena. Specific objectives of this paper are given below.

- Validation of finite element model developed by HyperForm using benchmark.
- Development of finite element model using HyperForm.
- Simulation of forming stages of wheel disc in order to study the thinning percentage, von Mises stress and formability.
- Implementation of optimization approach in order to control the excessive part thinning and springback phenomena.
- Simulation of further forming stages with optimized value of design variable.

V. VALIDATION OF FINITE ELEMENT MODEL DEVELOPED BY HYPERFORM

The first step in sheet metal forming numerical analysis involves development and validation of Finite Element model. The model developed needs to be validated using benchmark, which is a standard problem for which solution exist. The test problem for this work is taken from NAFEMS “Introduction to Non Linear Finite Element Analysis” edited by E. Hinton as reported in reference [6].



All dimensions are in mm
Fig 3: Schematic sketch of deep drawing technique

The test problem is illustrated in Fig 3 shows the deep drawing technique which is used to manufacture drink cans. As the deformation progresses by forcing the punch down the thin sheet makes contact with the punch and the die radius and draws in at the edges. The equilibrium path is non linear and higher order strains are induced.

Shell elements are used for the problem, which represents model parts that are relatively two-dimensional, such as sheet metal or a hollow plastic cowl or case and developed finite element model shown in the Fig 4.

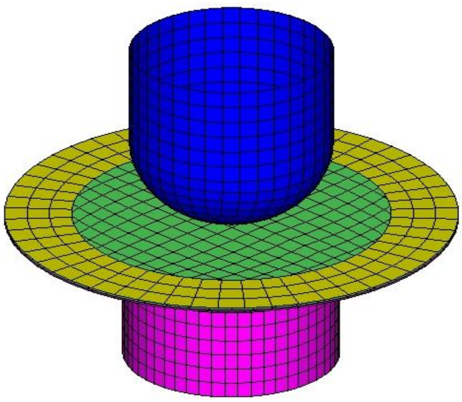


Figure 4: Finite element model of blank and tools

We can give the listed values of Table2 in HyperForm by invoking Elastic-Plastic material model /MAT/LAW43 (HILL_TAB), this law describes the Hill orthotropic material and is applicable only for shell elements.

Table 2: Material properties and mateial model for the test problem

Coefficient of friction, μ	0.04
Modulus of elasticity, E	112.688 GPa
Poisson's ratio, v	0.4068
Plastic deformation equations	For $0 \le \bar{\varepsilon}_p \le 0.36$ $\bar{\sigma}_y = 82.1 + 422.7\bar{\varepsilon}_p^{0.504}$ MPa n= 0.504 K= 422.7 For $\bar{\varepsilon}_p > 0.36$ $\bar{\sigma}_y = 82.1 + 371\bar{\varepsilon}_p^{0.375}$ MPa n= 0.375 K= 371 where n= Strain hardening exponent K= Strain hardening co-efficient(MPa)

Table 3: Boundary conditions applied for test problem

Binder Force $\mathbf{F_b}$	4904N (0.5 tons)
Binder Velocity	3000 mm/sec (negative Z direction)
Punch Velocity	3000 mm/sec (negative Z direction)

Target solution for the bench mark problem is the plot showing punch load v/s punch travel shown in Fig 5 and induced higher order strain is more than 30%. The results obtained from numerical simulation shown in Fig 6 and Fig 7 are in good agreement with the experimental results.

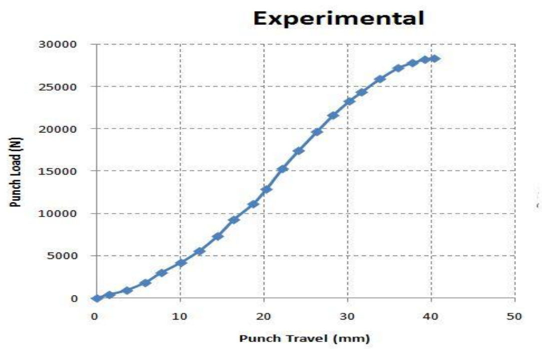


Figure 5: Experimental Result: Punch load v/s Punch travel plot

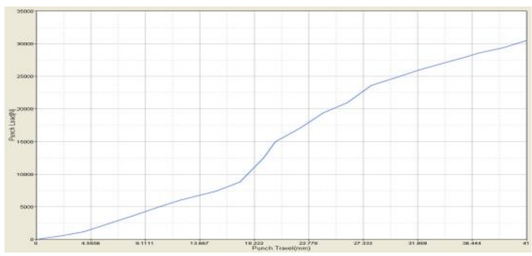


Figure 6: FEA result: Punch load v/s Punch travel plot

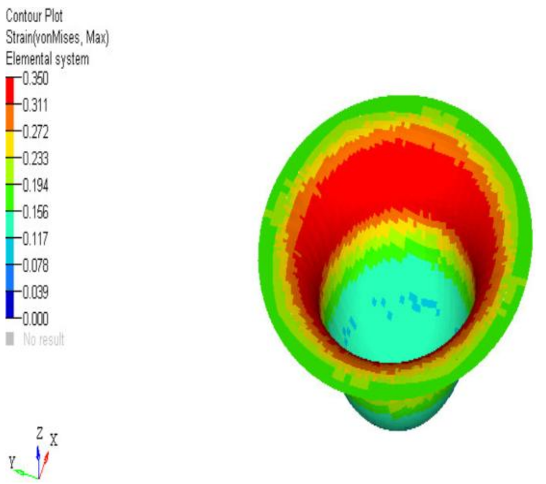


Figure 7: FEA result: Strain contour

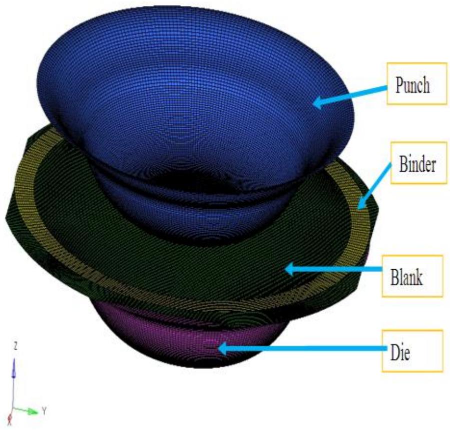


Fig 9: Finite Element Model for stage2

VI. CASE STUDY

a) Finite element model development

Since the wheel disc has a complicated shape it can't be formed in a single stage. Actual stages required to form wheel disc are identified and shown in the Fig 8, first step is to get the right blank shape, after this five forming steps follows, depending on how the tools will be designed. Finite element modeling, assigning material properties and boundary conditions has done using HyperForm.

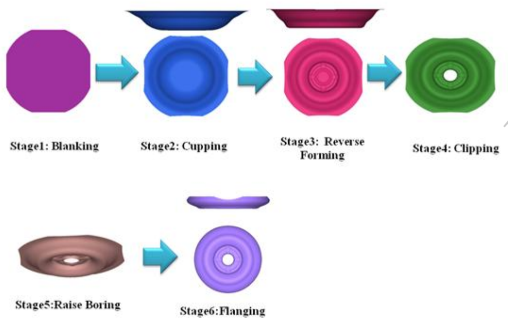


Fig 8: Forming stages of wheel disc

The mesh generation of blank needs to be done only in the first forming step since the blank from previous steps will be used for later forming steps and shell elements used for our problem. In the current work mesh generation for forming tools has done in each stage as per the dimensions and shape of the part to be formed. The finite element model for stage2 (cupping/ first forming) is shown in the fig 9. Boundary conditions applied are given in the Table4.

Table 4: Boundary conditions applied

Binder Force $\mathbf{F_b}$	235360 N (24tons)
Binder Velocity	5000 mm/sec (negative Z direction)
Punch Velocity	5000 mm/sec (negative Z direction)

b) Executing the simulation

Once the preprocessing is done, then simulation executed in Radioss. During the simulation, the results can be studied to detect problems in the simulation at an early stage. If the simulation of the first step looks good, one can continue with the second step. Doing everything stepwise will save lot of time, since unexpected problems often arise somewhere in the simulation.

c) Post-processing of stage2

The final stage in the simulation procedure is to evaluate and analyze the results. This has done in HyperView. Generally in sheet metal forming simulations, we are interested in evaluating the forming process by detecting cracks and wrinkles that would lead to failure.

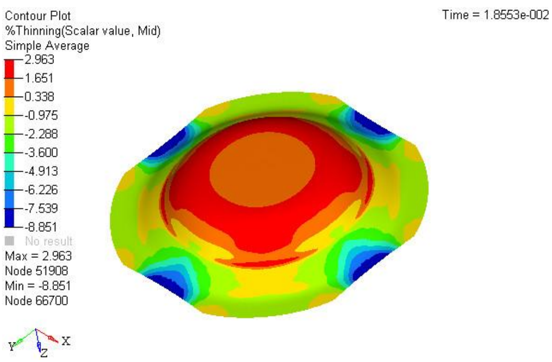


Fig 10: Stage2: Percentage thinning contour

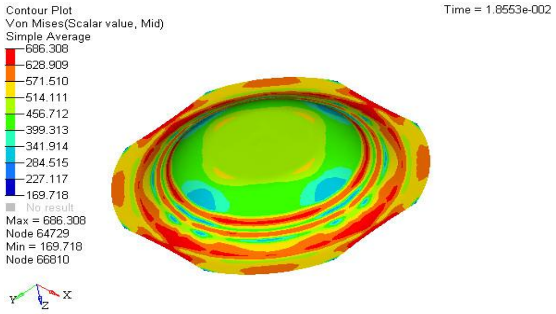


Fig 11: Stage2: von Mises stress contour

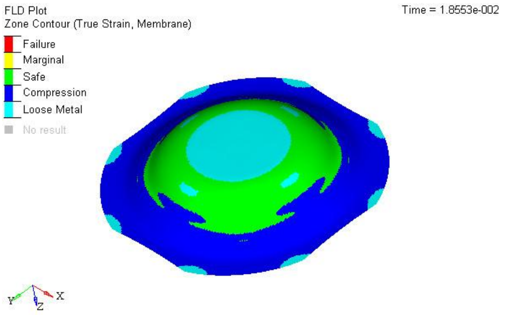


Fig 12: Stage2: FLD zone contour

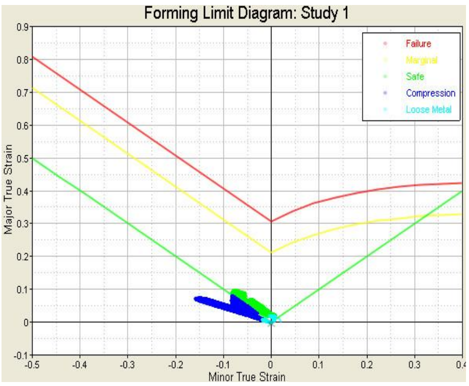


Fig 13: Stage2: Forming Limiting diagram

d) Result interpretation of stage2

The maximum percentage thinning observed for stage2 is 2.9% as shown in the Fig 10 .Maximum true stress of 686.3 MPa is observed from the Fig 11, which is under the ultimate true stress of 701.5 MPa. Since there are no elements in failure region, from the Formability limiting diagrams shown in the Fig 12 and Fig 13 shows that the component is safe, compression zone occuring in the flange region and tension in the cupping region where the material is drawn inside.

e) Forming simulation of stage3 (Reverse Forming)

In this step we will use the formed part of the previous stage as initial blank. The initial blank for stage3 is as shown in Fig 14, which is the formed part of stage2 and complete finite model for stge3 is shown in Fig 15. The simulation methodology and boundary conditions are same as applied for stage2.

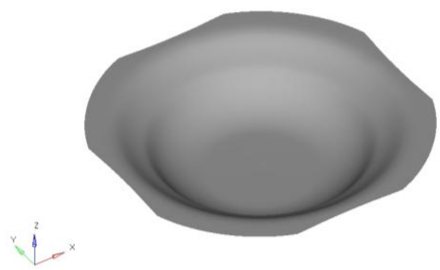


Fig 14 : Formed part of stage2 or initial blank for stage3

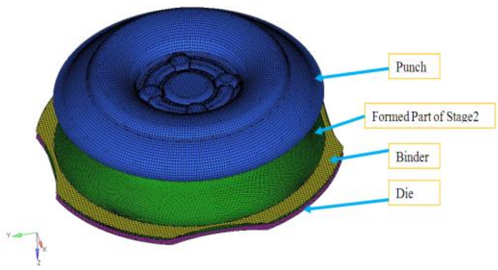


Fig 15: Finite element model for stage3 tools

f) Post-processing of stage3

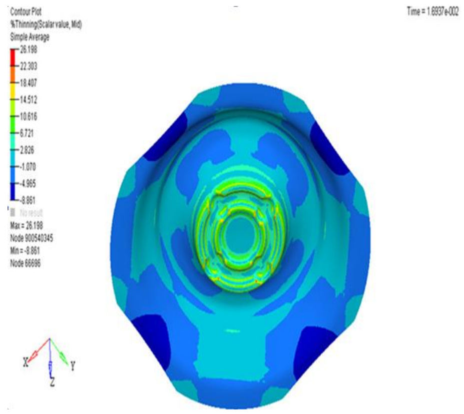


Fig 16: Percentage thinning contour for stage3

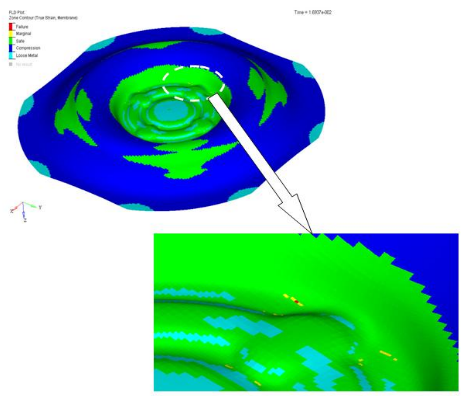


Fig17: FLD zone counter showing elements at failure zone

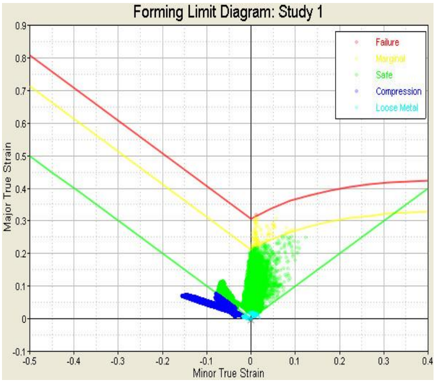


Fig 18: Formability limiting diagram showing elements at failure zone

Generally in most industrial applications for steels and its alloys the maximum allowable thinning is 20%, from the Fig 16, we can observe that maximum thinning is 26.198% and the regions with excessive thinning are prone to the crack initiation and finally to the failure of a component. From FLD plots shown in Fig 17 and Fig 18 we can see the elements in failure zone.

Now our main goal is to prevent the excessive thinning without changing the design of the component.

g) Proposed optimization approach

The main aim of optimization is the prevention of excessive part thinning but one more factor we need to keep in our mind is springback effect therefore before going for the optimization of the forming process we need to estimate the spring back also. Excessive thinning prevention problems are typically multi-objective ones since springback effect has to be managed too.The blank holder force (BHF) value was considered as process design variable. Proposed optimization is the general method for to control excessive thinning and spring back effect.

The proposed approach is deterministic approach in which process variability due to noise factors are completely neglected and therefore only blank holder force(BHF) influence was considered. Variations of coefficient of friction ‘μ’ and strain hardening exponent ‘n’ are neglected and values of μ and n are given in the Table 1. The deterministic procedure steps are as follows.

1. Design of experiment (DOE) definition

Since one single input parameter was selected, BHF value is reduced from initial 24Ton to 4Ton with the constant decrement of 2Ton.

2. FEM simulation and results data collection

A numerical simulation for each DOE point was run and thinning percentage and springback results were collected are shown in the Table4.3.We already have the thinning percentage but we need to calculate the springback for 24Ton BHF and other DOEs.

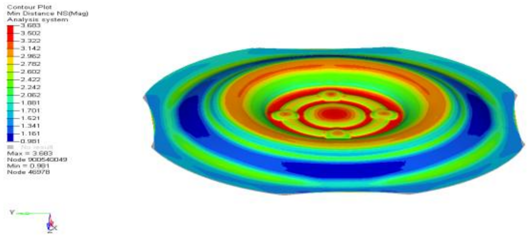


Fig 19 : Distance plot for springback calculation

As springback is concerned, a CAD environment was utilized in order to provide a good evaluation of springback. In particular, comparisons between deformed blank after load removing and final stamped part were developed trying to measure different springback indicators.

HyperForm gives the distance plot between die and formed part as shown in Fig 19, then we need to export the necessary nodal values in Microsoft Excel and average value is taken as the springback measure, the same procedure is repeated for all DOEs.

we need to follow the same procedure followed for the stage3 forming simulation to get thinning percentage and springback values for remaining DOFs.

Table 4.3: FEM simulation outputs

DOE	BHF in tons	%THINNING	SPRING BACK (mm)
1	24	26.198	0.16
2	22	24.6	0.18
3	20	22.3	0.21
4	18	19.32	0.30
5	16	18.5	0.44
6	14	17.68	0.50
7	12	15.6	0.62
8	10	15.1	0.66
9	8	14.32	0.67
10	6	14.02	0.68
11	4	13.32	0.69

3. Regression model development:

Second Order regression model was developed to formulate surface response (y) in terms of each objective function and then calculated regression coefficient which indicates the best fit of the approximated curves. Fig 20 and Fig 21 shows the fitting the curves and we observe that both curves present the monotonous behavior with respect to the BHF changes.

Microsoft Office Excel Worksheet is used to generate the curves. In particular, a stepwise regression was developed by eliminating, progressively, the less statistically significant terms and trying to optimize the correlation index R^2 (adj), which provided a measure of model approximation capability.

$$y\,(BHF) = 0.0285x^2 - 0.1513x + 13.672$$

Where x = %thinning

$$R^2 = 0.9918\ (99.18\%)$$

The value of R^2 (adj) higher than 99.18% were reached, confirming a good data approximation.

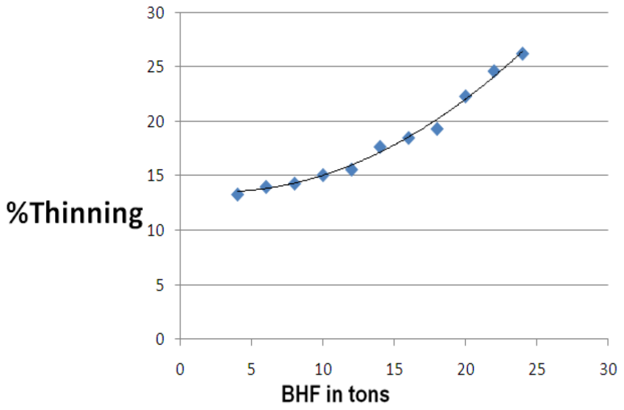


Fig 20: %Thinning v/s BHF

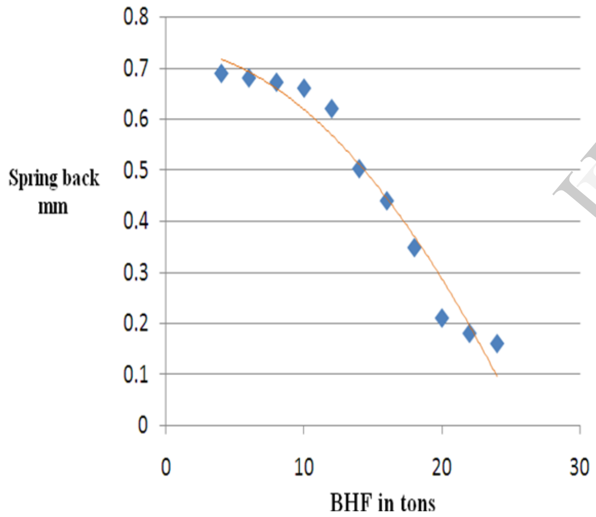


Fig 21: Spring back v/s BHF

$$y\,(BHF) = -0.001x^2 - 0.0017x + 0.7406$$

Where x = Springback

$$R^2 = 0.9642(96.42\%)$$

The value of R^2 (adj) higher than 96.42% were reached, confirming a good data approximation.

4. Pareto frontier

Once the two objective functions were analytically formulated, it was necessary to identify compromise solutions minimizing simultaneously %thinning and Springback. Therefore, a Pareto frontier was constructed, which helps to identify the compromise solution minimizing simultaneously thinning and springback. In fact, every %thinning and springback couple corresponding to a particular BHF value is a Pareto solution: no solution exists that (for given operative conditions, i.e. for given BHF) for fixed thinning allows a lower springback. Fig 22 shows the obtained Pareto frontier.

The maximum thinning increases as springback decreases in the stamped part. Pareto frontier is a very useful design tool, since it allows visualizing all the possible compromise solutions. Pareto frontier makes possible to predict the best possible value of a given objective function once a value of the other one is fixed: if a thinning of about 19.32% (point **S** in Fig 22) is desired, the best possible value of springback to be expected is given by the Pareto frontier (0.34mm in the case of point S).

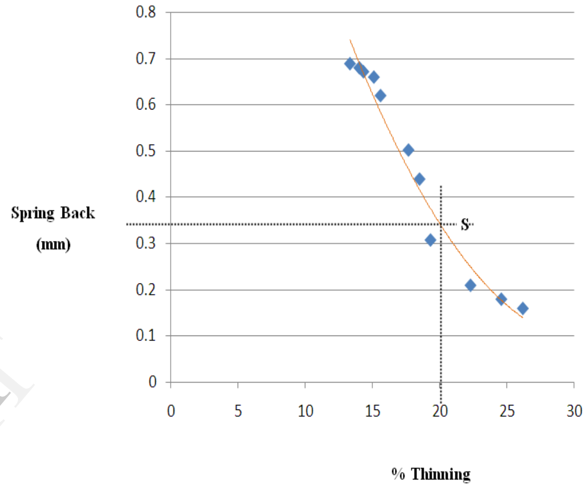


Fig 22: Pareto frontier

Now the optimal value for the BHF is 18 Ton in order to have compromising values of thinning percentage and Spring back value. Now we need to use BHF 18 Ton for stage3 simulation and next forming simulation of Wheel Disc.

h) Stage3 simulation results for optimized value of BHF

All the boundary conditions applied will remain same except only BHF ie, 18 Ton applied.

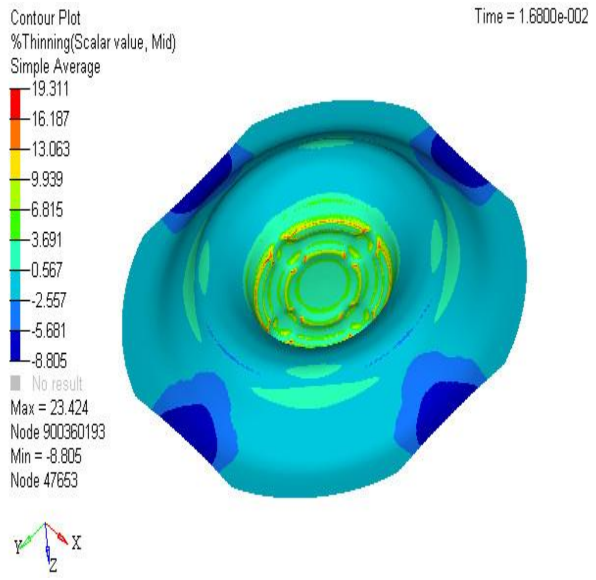


Fig 23: Thinning percentage contour for stage3 (18 Ton)

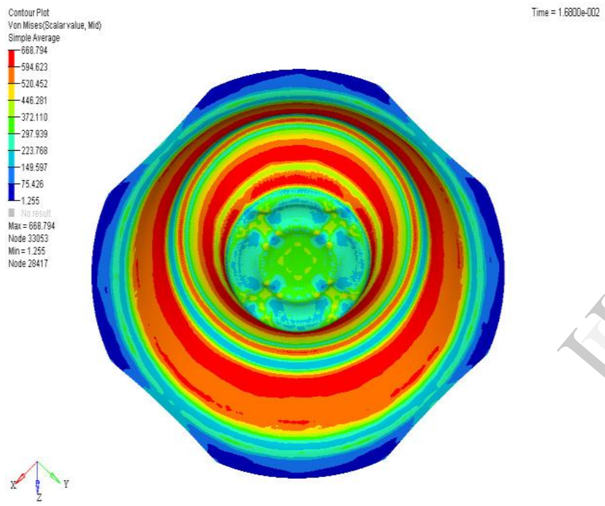


Fig 24: von Mises stress contour for stage3 (18 Ton)

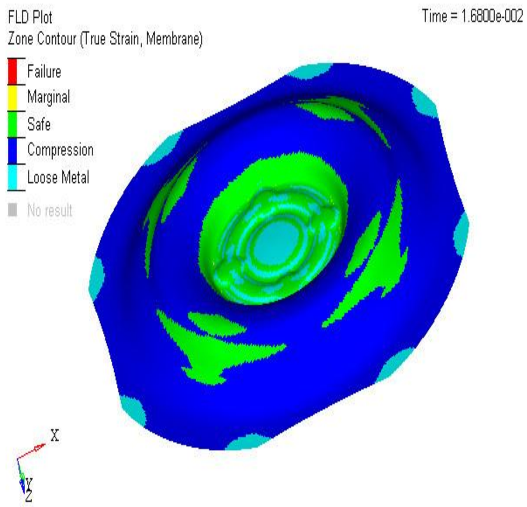


Figure 25: FLD zone contour for stage3 (18 Ton)

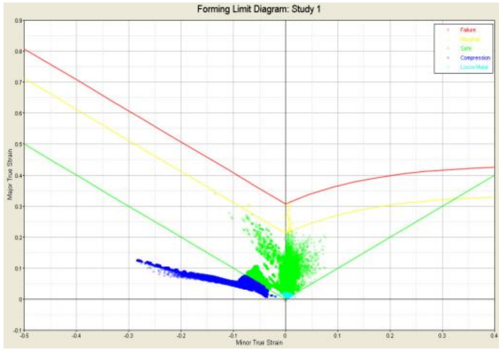


Fig 26: Formability limiting diagram for stage3 (18 Ton)

i) Result interpretation of stage3

From Fig 23 for BHF value 18 Ton the percentage thinning is 19.3%, which is less than as observed for 24Ton. Maximum true stress of 668.7 MPa observed in the Fig 24 is less than ultimate true stress 701.5 MPa and from Figure 25 and Fig 26, hence we can observe that the component is safe.

j) Stage 4: Inner clipping

Forming in Stage4 involves circular clipping of the component at the centre as shown in Fig 27. This is achieved by pre-processing operations. Since clipping is a preprocessing operation, stresses induced here will be carried over to the stage 5.

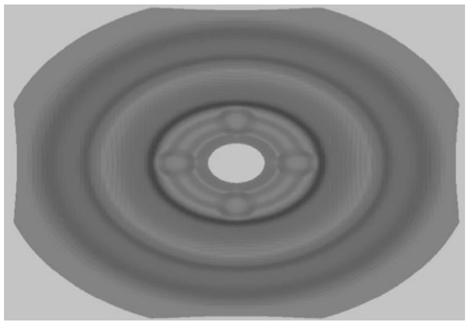


Figure 27: Inner clipping

k) Stage 5: Raise boring

Raise boring set up is shown in the Fig 28 and diameter of the hole at the centre has increased and extreme edge at the hole is bent to 90^0.

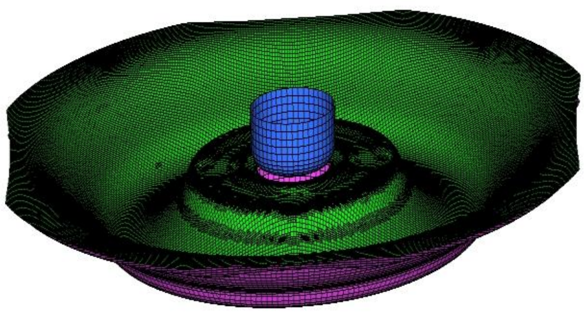


Figure 28: Finite element model for stage5

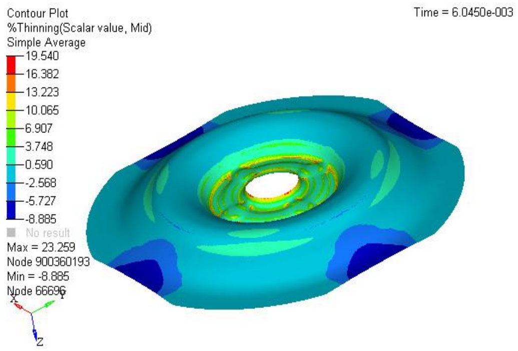


Figure 29: Thinning contour for stage5

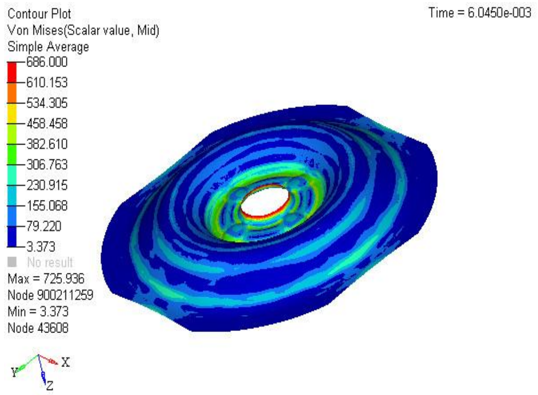


Fig 30: von Mises stress contour of Raise boring

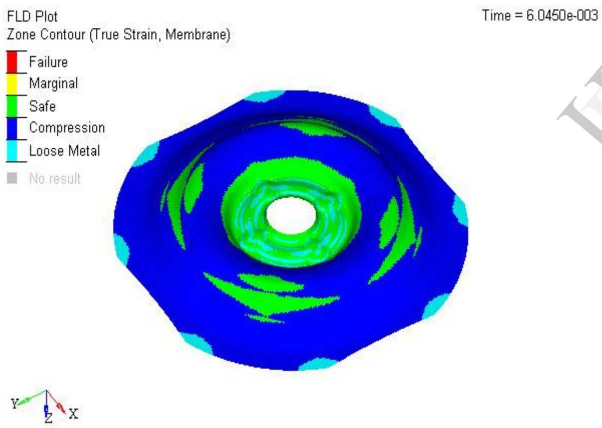


Fig 31: FLD zone contour for stage5

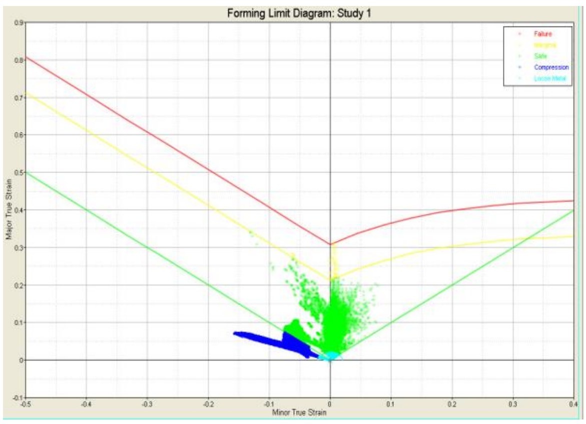


Fig 32: Forming limiting diagram for stage5

l) Result interpretation of stage5

From von Mises contour and FLD plots shown in the Fig 30 and Fig 32 respectively ,we can observe that component is safe.

m) Stage6: Flanging

Developed finite element model for flanging is shown in Fig 33.

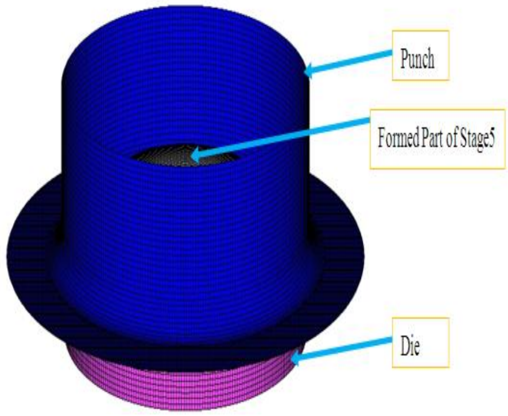


Figure 33: Finite element model for stage6

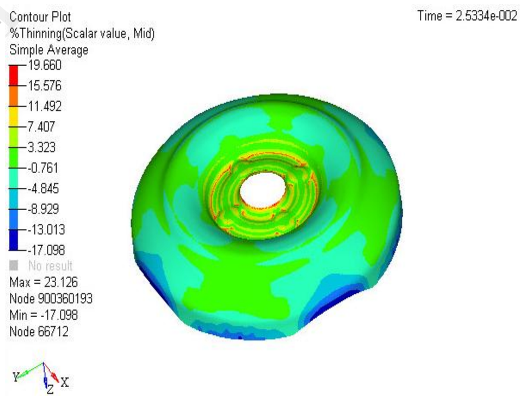


Figure 34: Thinning percentage contour for stage6

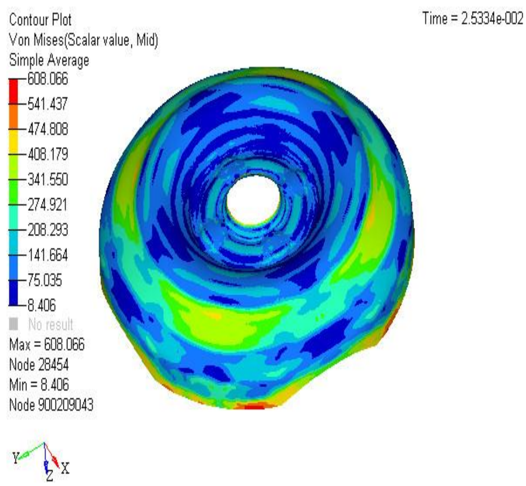


Figure 35: von Mises stress contour for stage6

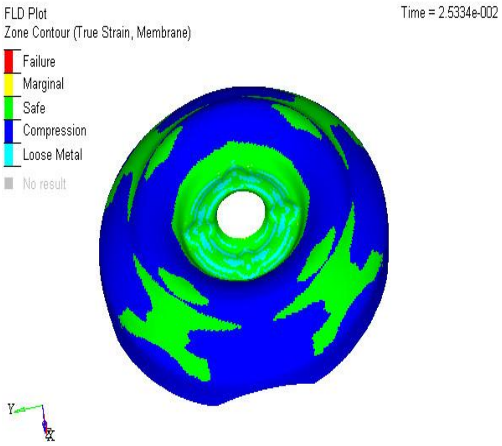


Figure 36: FLD zone contour for stage6

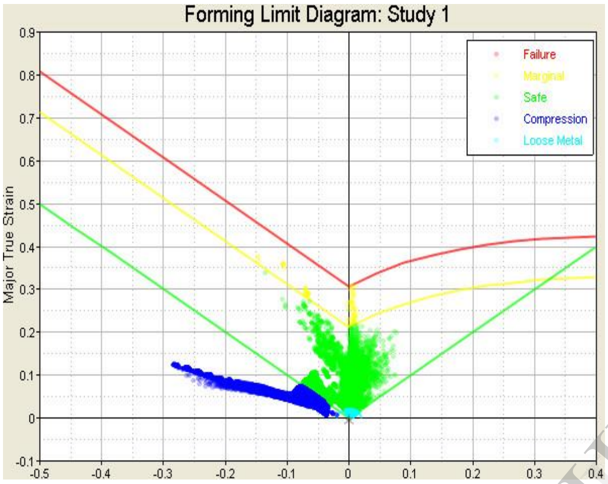


Figure 37: Formability limiting diagram for stage6

n) Result interpretation of stage6 (final stage)

A maximum true stress of 608MPa observed in Figure 34, which is under the ultimate true stress of 701.5MPa and also there are no elements at the failure zone as observed from Formability limiting diagram shown in Figure 36 and Figure 37, hence we observed that the final formed wheel disc is safe.

VII. CONCLUSION

Manufacturing of complex components in multi steps like wheel disc demands numerical simulation to arrive at satisfactory tools and to identify the processing parameters involved in the forming process. FE Modeling and Numerical simulation using commercial FEA software (HyperForm) is demonstrated in the present work.

Multi step forming simulation gives the clear idea of excessive thinning, magnitude of stress, thickness variation in the formed part at each stage and then we can identify the problem in the design stage and take the necessary action against that.

In this work excessive thinning observed in the stage3 (reverse forming) of wheel disc and at the same time we considered the springback effect also.

In the present work Multi Objective Optimization approach has proposed in order to prevent the excessive thinning and spring back effect. The proposed optimization approach is an integration of finite element method and surface response method. With the optimal value of design variable we continued the forming steps and brought down the excessive thinning.

REFERENCES

1. Y. Huang Z.Y. Lo and R.Du, *Minimization of the thickness variation in multi-step sheet metal stamping*, Journal of Materials Processing Technology, Vol. 177, 2006, pp 84-86.
2. L. Marretta, G. Ingarao and R. Di Lorenzo, *Design of sheet stamping operations to control springback and thinning: A multi-objective optimization approach*, International Journal of Mechanical Sciences, Vol. 52, 2010, pp 914-927.
3. Sergey F. Golovashchenko, Nicholas M. Bessonov and Andrey M. Ilinich, *Two-step method of forming complex shapes from sheet metal*, Journal of Materials Processing Technology, vol. 211, 2011), pp 875-885.
4. T. Cwiekala, A. Brosius and A.E. Tekkaya, *Accurate deep drawing simulation by combining analytical approaches*, International Journal of Mechanical Sciences, Vol. 53, 2011, pp 374-386.
5. B.T. Tang, Z. Zhao, X.Y. Lu, Z.Q. Wang, X.W. Zhao, S.Y. Chen, *Fast thickness prediction and blank design in sheet metal forming based on an enhanced inverse analysis method*, International Journal of Mechanical Sciences, Vol. 49, 2007, pp 1018-1028.
6. NAFEMS, *Introduction to Non Linear Finite Element Analysis*, edited by E. Hinton.
7. ALTAIR HyperWorks manufacturing solutions help guide.
8. ALTAIR Radioss help guide.