

# Finite Element Analysis and Optimization of Weld Distortion in Automobile Chassis

Akshay Nighot<sup>1</sup>, Prof. Anurag A. Nema<sup>2</sup>, Prof. Anantharma A<sup>3</sup>

<sup>1</sup>M.E. Student Dhole Patil College of Engineering, Pune, Maharashtra

<sup>2</sup>Assistant Professor Dhole Patil College of Engineering, Pune, Maharashtra

<sup>3</sup>Assistant Professor Dhole Patil College of Engineering, Pune, Maharashtra

\*\*\*

**Abstract** - Chassis is the critical component, the 'backbone' of any automobile and it is on this that various components come and assembly. To facilitate this assembly the chassis has several bracket welded to it. During welding, the process is not properly controlled, and then it may result in excessive weld distortion or residual stress, both of which harm the long term life of the chassis. Hence it is imperative that weld distortion be minimized. In this case the chassis assembly is to be analyzed for the existing weld sequence, and check deficiencies in the process. Then based on results an improvised sequence is to be recommended. The objectives of the project are, to create analysis of standard operating procedure in ANSYS and to study the weld distortion in current weld structure and then optimize the welding along weld sequence, weld duration and clamp location respectively.

**Key Words:** backbone, welds distortion, weld sequence, weld duration, and clamp location.

## 1. INTRODUCTION

Welding is widely used in automotive industries to assemble various products. The chassis is considered to be one of the significant structures of an automobile. It is the frame which holds both the car body and the power train. When structures are manufactured by welding, a non-uniform temperature distribution is produced. This distribution initially causes a rapid thermal expansion followed by a thermal contraction in the weld and surrounding areas, thus generating inhomogeneous plastic deformation and residual stresses in the weldment when it is cooled. Thus, it is important to evaluate the deformation due to welding. FEA is most dominant method for calculation and simulation of computer made models. The name FEA comes from the way a complicated model is divided into a model that is built by small elements. The traditional manufacturing processes were mainly established from trial-and-error experiments approach. Such trial-and-error procedure approach requires tremendous material, energy, labors, as well as procedure significant waste, fumes and emissions. The appropriate control techniques are mandatory with reliability and cost effectiveness for the application of

welding process. The welding simulations simulate an actual welding process based on science and physics and the tests can be performed inside computer without wastage of resources and hazardous environment impact. A hybrid approach involving both FE modeling and experimental work has proven very beneficial.

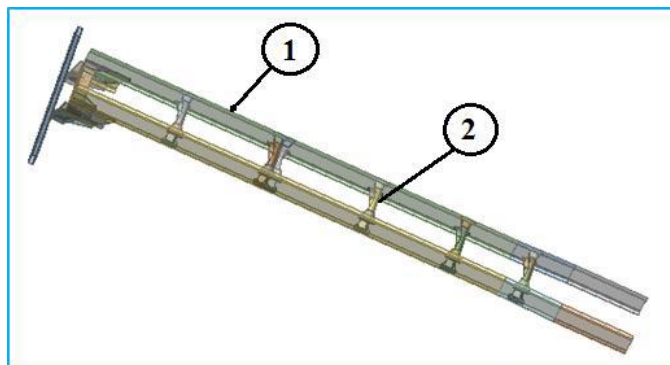
## 2. LITERATURE REVIEW

Chang Doo Jang , Yong Tae Kim , Young Chun Jo , Hyun Su Ryu [1] Welding deformation reduces the dimensional accuracy of ship hull blocks and decreases productivity due to the correction work. Prediction and minimizing of welding distortion at the design stage will lead to higher quality as well as higher productivity. Dragi Stamenković (2009)[2] This work presents the FE model for numerical simulation of welding residual stresses in high strength carbon steel butt welds. The finite element method is an efficient technique in analysing residual stresses in welding processes. Andrea Capriccioli (2009)[3] In order to reduce the computer time a simplified procedure has been developed in the case of TIG welding mechanical analysis: this procedure lets to run the whole welding mechanical simulation with a considerable saving in computer time. Nuraini, A.A. (2013)[4] The results shown that the optimum welding parameter able to lower the temperature and residual stress of the weld center area compare to non-optimum parameters around 8 percent for temperature and 1.5 percent for residual stress. E. Armentani (2007)[5] They studied as transient temperature distribution in a welded joint is highly affected by thermal conductivity. Large tensile longitudinal residual stresses are present near the welding bead. S. R. Patil, C. A. Waghmare [6] presents the influence of welding parameters like welding current, welding voltage, welding speed on ultimate tensile strength (UTS) of AISI 1030 mild steel material during welding. From this study, it is observed that welding current and welding speed are major parameters which influence on the tensile strength of welded joint. Chattopadhyay, G. Glinka, M. El-Zein, J. Qian, R Forams [7] Fatigue analyses of weldments require detailed knowledge of the stress fields in critical regions. C.M.Cheng, C.P.Chouy, I.K.Lee and H.Y.Lin [8] In this study they conducted single V-groove butt welding (GTAW) on three types of heat treatable aluminium alloys

2024-T351, 6061-T6 and 7075-T6 and compared the angular distortion levels of the three aluminium alloys at different Vee preparation angles with or without restraint. Y. P. YANG, R. DULL, H. CASTNER, T. D. HUANG, AND D. FANGUY [9]

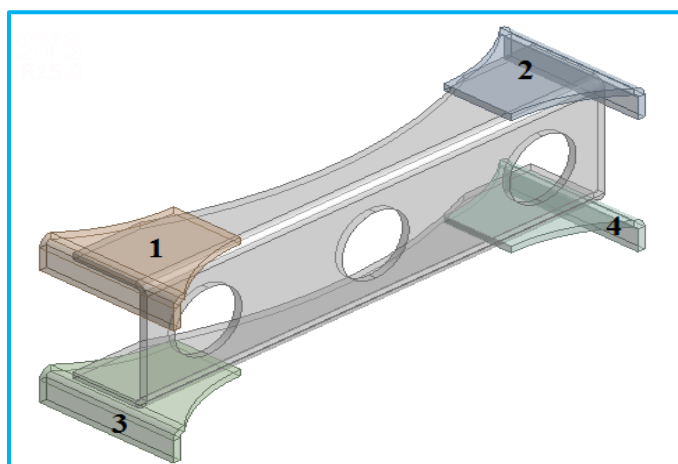
### 3. PROBLEM DEFINITION

Chassis is the critical component, the 'backbone' of any automobile and it is on this that various components come and assembly. To facilitate this assembly the chassis has several bracket welded to it. During welding, the process is not properly controlled, and then it may result in excessive weld distortion or residual stress, both of which harm the long term life of the chassis. Hence it is imperative that weld distortion be minimized. In this project the chassis assembly is to be analysed for the existing weld sequence, and check deficiencies in the process. Then based on results an improvised sequence is to be recommended.



**Figure-1** Geometric CAD model of commercial vehicle chassis showing 1- C channel, 2- Rib assembly.

Welding process is carried out in between plates present on rib six rib assemblies and respective C- channels. There are total 24 numbers of plates present in model.



**Figure-2** Shows Rib assembly and 1,2,3,4 are plates present in assembly

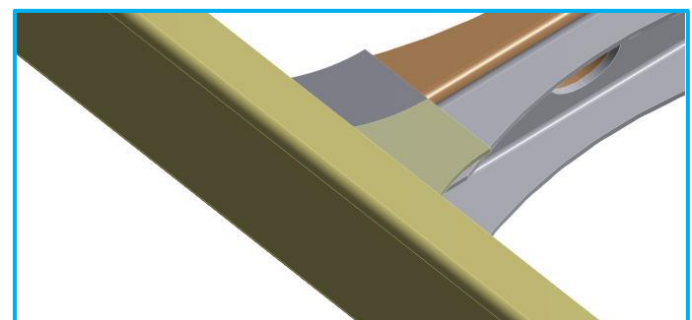
### 4. METHODOLOGY

The proposed work involves the following steps:

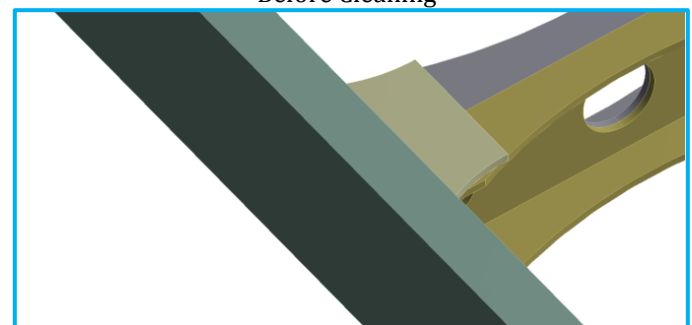
1. Perform cleaning operation of the model which is helpful in development of proper meshing. And develop proper method for meshing.
2. Perform preliminary analysis with half model of chassis with reduced number of nodes and hence the solution time, to check out the result pattern and to develop standard operating procedure in workbench.
3. Considering full model of chassis perform Transient Thermal analysis with given input condition.
4. Perform Transient Structural Analysis with result obtained in thermal analysis as an input named imported body temperature along with the fixed supports to calculate deformation.
5. Perform above steps with different weld path's to calculate minimum deformation and hence the optimum weld path.
6. Calculate the distortion and residual stresses for the optimum path.
7. Validation of results will be done by comparing distortion at optimum weld path with same path at designated working conditions.

### 5. CLEANING OF MODEL

Much of the hard work in setting up an analysis is related to creating the desired mesh. However; complicated geometry can make this a difficult task. Repair utilities that can be used to clean up complicated CAD geometry. These simplifications, the various modelling operations are available in Design Modeller.



Before Cleaning



After Cleaning

**Figure-3** Before and after cleaning of same part

## 6. MESHING OF MODEL

**Table -1:** Mesh control of a model (\* Size varies as per number of nodes)

Element Type	SOLID 186
Method of mesh control	Hex Dominant
size	25*
Statistics	
No. of nodes	1,82,207
No. of elements	28,421

## 7. PRELIMINARY ANALYSIS WITH HALF CHASSIS

There are mainly four reasons behind the division of model into half, they are as follows,

- Development of standard operating procedure for analysis.
- Computer system limitation.
- Increased number of mesh nodes and elements.
- Increased solution time.

### 7.1 TRANSIENT THERMAL ANALYSIS

#### Input Parameters

- Type of welding = MIG welding.
- Maximum Temperature reached during welding = approximately 3,000 °C.
- Welding Speed = 12 inches/min= 5.08 mm/sec.
- Convection film coefficient = 0.5 w/m<sup>2</sup>.
- Flux required = 5.2w/mm<sup>2</sup>.

#### Time Calculation

Welding time calculation for first four plates of model of chassis are shown,

$$\text{Welding speed} = \frac{\text{Length of weld pool}}{\text{Arc on Time}}$$

Here, length of weld pool = 445mm

$$\text{Arc on Time} = \frac{445}{5.08} \text{ sec}$$

$$\approx 90 \text{ sec}$$

For example, transient thermal analysis for first four plates is carried out in three numbers of steps. First two steps involve welding operation and third step dedicated to cooling which equals to 90 seconds. Each step carries time equal to 90 seconds according to above calculation. Step number three ends at 270 second.

**Table -2:** Welding Time required for each plate (included in full chassis also).

Sr. No.	Plate Number	Welding Time (in second)
1	1 to 4	90
2	5 to 8	60
3	9 to 12	80
4	13 to 16	60
5	17 to 20	60
6	21 to 24	60

Analysis starts with applying different boundary conditions to a given model basically boundary condition contains,

- Provision of convection
- Provision of heat flux

#### Result For Transient Thermal Analysis



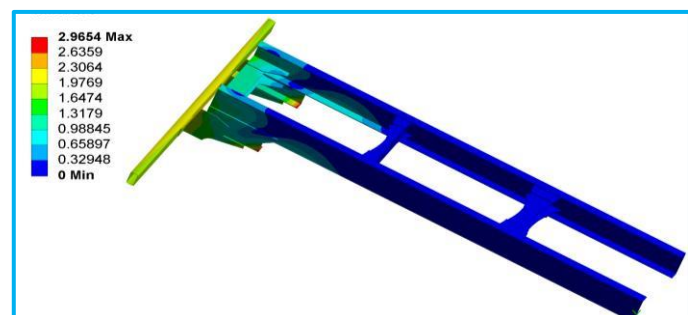
**Figure-4** Temperature output for first four plates in Transient thermal analysis is 2620°C.

### 7.2 TRANSIENT STRUCTURAL ANALYSIS

#### Input Parameter

- Result of transient thermal analysis
- Source time and analysis time

#### Result of Transient Structural Analysis



**Figure-5** Deformation for first four plates in Transient Structural analysis is 2.9654 mm.

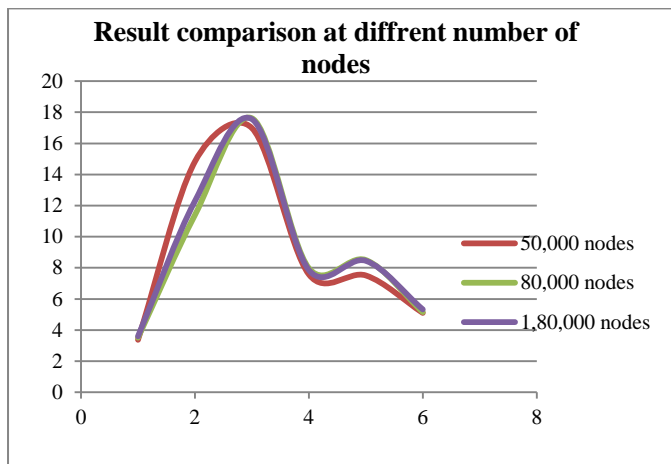
From the analysis of half model of chassis standard operating procedure is properly developed to calculate the final deformation.

### 8. Analysis of Full Model of Chassis

Analyses full model of chassis is analysed for three different cases of mesh sizes described as follows,

- Approximately 50,000 nodes.
- Approximately 80,000 nodes.
- Approximately 1,80,000 nodes.

**Note:** It is difficult to achieve exact number of nodes in Ansys therefore approximate values are taken under consideration.



**Chart -1** Graph of deformation vs. no. of plate

From the graph of number of plates vs. deformation, it is concluded that the results for different mesh size are approximately same and variation is within the considerable limit. Hence further analysis can be done on approximately 50,000 nodes, as it reduces the solution time and the results are approximately same compared to fine mesh.

### 9. Weld Path Optimization

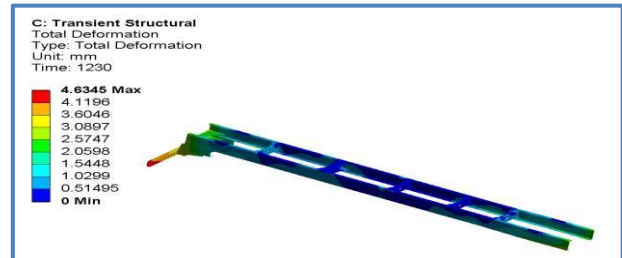
Analysis are done with full chassis by using different weld path to achieve minimum deformation, those cases are,

- **Path 1** linear application of heat flux at all plates serially throughout the length.
- **Path 2** alternate application of heat flux to the rows of plates throughout the length.
- **Path3** application of heat flux to the top plates of consecutive rows simultaneously and then respective bottom plates simultaneously. In this case 4 plates getting weld at a time. In this case 4 plates getting weld at a time.
- **Path 4** simultaneous application of heat flux to the top plates then bottom plates of alternate rows throughout the length.

According to the weld path discussed above analysis are carried out at number of nodes equal to 50,000 (approximately) and results are discussed as below,

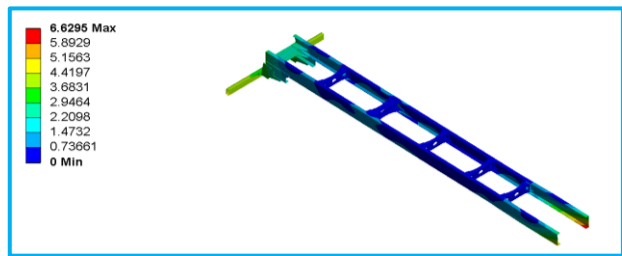
## 10. RESULTS

### 10.1 Results of Path 1



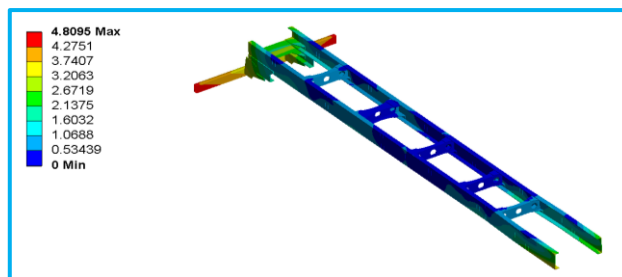
**Figure-6** Deformation due to Transient Structural analysis for weld path 1 is 4.6345 mm.

### 10.2 Results of Path 2



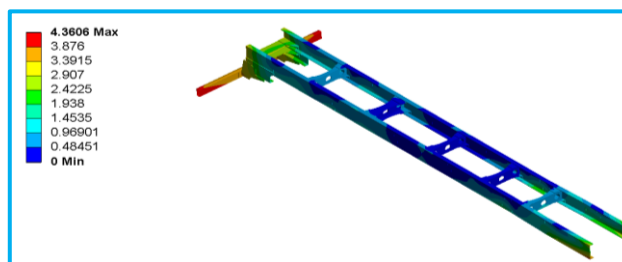
**Figure-7** Deformation due to Transient Structural analysis for weld path2 is 6.6295 mm.

### 10.3 Result of Path 3



**Figure-8** Deformation due to transient structural analysis for weld path 3 is 4.8095 mm.

### 10.4 Result of Path 4



**Figure-9** Deformation due to transient structural analysis for weld path 4 is 4.3606 mm.



**Table-3** Results of analysis in tabular form.

Sr. no	Weld Path	Deformation(mm)
1	Weld Path 1	4.3645
2	Weld Path 2	6.6295
3	Weld Path 3	4.8095
4	Weld Path 4	4.3606

From the above results, it is seen that Weld Path 4 has minimum deformation of 4.3606 mm which is minimum of all the cases stated above, thus weld path 4 (Simultaneous application of heat flux to the top and bottom plates of alternate rows throughout the length) is considered as the optimum path for welding for given model of chassis.

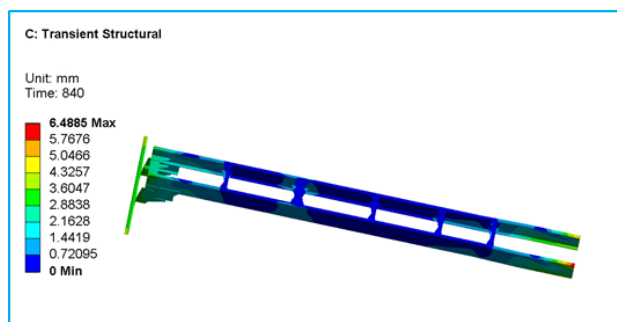
### 11. Distortion due to Welding in Optimized Case

Welding distortion, a result of the non-uniform expansion and contraction of weld metal and adjacent base metal during the heating and cooling cycle of the welding process, is a major concern during the fabrication of a welded structure.

Distortion is nothing but the deformation beyond yield strength, so considering the following properties in further analysis to calculate Distortion,

- Yield Strength = 2.5E+08 Pa
- Tangent Modulus = 1.45E+09 Pa

#### 11.1 Result For Transient structural Analysis



**Figure-10** Distortion due to transient structural analysis for optimum weld path is 6.4885 mm

### 12. Conclusions

1. The Project basically focused on an Analysis and Optimization of weld distortion in automobile chassis. The analysis of chassis model was done in ANSYS 15.0 Workbench. The results were supported with an experimental validation for verifying the actual distortion and FEA results. Following are concluding

remarks based on the analysis performed on chassis model.

2. Firstly analysis of half model is done to develop the standard operating procedure. From the comparison of results at different mesh sizes it is concluded that variation in results is within acceptable limit, hence approximately 50,000 nodes mesh size is fixed for further analysis.
3. Full model of chassis is analyzed under 4 different weld paths to find out the minimum distortion, and hence optimum weld path (Simultaneous application of heat flux to the top and bottom plates of alternate rows throughout the length) is decided. Optimum weld path has minimum distortion of 6.4885 mm

FEA results and Experimental results are in close resemblance and proved that the optimum weld path decided by FEA analysis is correct and is validated by experimental distortion results is 6.8 mm. % error between FEA and experimental result's is 4.5% which is less than the allowable error (10%) in FEA.

From the above discussion it is concluded that FEA result is reliable and FEA is an ideal tool for such nonstandard situations and also for research work.

### ACKNOWLEDGEMENT

Thanks to Mr. Vinay Patil for his valuable contribution in developing the research work.

### 13. References

- [1]. S. R. Patil, C. A. Waghmare, "Optimization of MIG welding parameters for improving strength of welded joints" E-ISSN2249-8974 Int. J. Adv. Engg. Res. Studies / II/ IV/July-Sept., 2013/14-16
- [2]. A .Chattopadhyay, G. Glinka, M. El-Zein, J. Qian, R Forams, "STRESS ANALYSIS and FATIGUE of welded structures, 1106\_0037\_WELDING\_7\_8\_2011.indd 20
- [3]. Chang Doo Jang, Yong Tae Kim, Young Chun Jo, Hyun Su Ryu, "Welding Distortion Analysis of Hull Blocks Using Equivalent Load Method Based on Inherent Strain", 10th International Symposium on Practical Design of Ships and Other Floating Structures Houston, Texas, United States of America © 2007 American Bureau of Shipping.
- [4]. C.M.Cheng, C.P.Chouy, I.K.Lee and H.Y.Lin, "Distortion Analysis of Single V-groove Butt Welding on Heat Treatable Aluminium Alloys", J. Mater. Sci. Technol., Vol.21 No.5, 2005

[5]. Y. P. Yang, R. Dull, H. Castner, T. D. Huang, and D. Fanguy, "Material Strength Effect on Weld Shrinkage and Distortion", NOVEMBER 2014 / WELDING JOURNAL 421-s

[6]. N. R. Mandal and C. V. N. Sundar, "Analysis of Welding Shrinkage", WELDING RESEARCH SUPPLEMENT I 237-s JUNE 1997

[7]. Z. M. Liu, C. S. Wu, Y. K. Liu, and Z. Luo, "Keyhole Behaviour's Influence Weld Defects in Plasma Arc Welding Process", SEPTEMBER 2015 / WELDING JOURNAL 281-s

[8]. Dean Deng, Ninshu Ma, and Hidekazu Murakawa, "Finite Element Analysis of Welding Distortion in a Large Thin-plate Panel Structure", Transactions of JWRI, Vol.40 (2011), No. 1

[9]. Nuraini, A.A, Zainal A.S.M. and Azmah Hanim, M.A, "The Effect of Welding Process Parameter on Temperature and Residual Stress in Butt-Joint Weld of Robotic Gas Metal Arc Welding", Australian Journal of Basic and Applied Sciences, 7(7): 814-820, 2013 ISSN 1991-8178

[10]. Z. Barsoum, "Determination of Weld Loads and Throat Requirements Using Finite Element Analysis with Shell Element Models--A Comparison with Classical Analysis", Welding research supplement April 1999

[11]. Dragi Stamenkovic, Ivana Vasovic, "Finite Element Analysis of Residual Stress in Butt Welding Two Similar Plates", Scientific Technical Review, Vol.LIX, No.1, 2009

[12]. K.T. B. Padal, K. Ajay Kumar, "Optimization and Control of Welding Distortion in Hatch Cover Girders", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 11, Issue 2 Ver. VII (Mar- Apr. 2014), PP 22-29