International Engineering Research Journal

Optimization of Commercial Load Body

Pushpak J. Joshi†, S. B. Patil‡
†Mechanical Engineering Department, SCOE, SavitribaiPhule Pune University, Maharashtra, India.
‡Mechanical Engineering Department, SCOE, SavitribaiPhule Pune University, Maharashtra, India.

Abstract

The structural components of the vehicle and the vehicle itself, are subjected to loads which cause stresses, deflections, strains, vibration and noise in the components.To achieve a quality vehicle, i.e. one having reduced weight, and reduced cost and so on, it becomes necessary to use materials of appropriate strength and stiffness property with the most appropriate geometry. This research addresses responses of a vehicle when stich welding is used between C-channel (cross member) and it's bracing using rigid elements (line elements). The entire analysis is done for Tata Motors HCV load carrier. The stresses and deformations generated on the load body are compared in both full and stitch weld conditions. In this manner, we have optimized the welding for cross-members of load carrier thereby reducing the manufacturing cost with increased productivity.

Keywords: *Load Body, Stitch weld, Intermittent weld, Heavy Commercial Vehicle, Optimization.*

1. Introduction

Load Body is the part of the commercial vehicle which takes the payload. So the analysis of this part becomes critical. The aim of this project is to determine the stresses in this structure due to various loading conditions. The loading conditions involves with payload and without payload. It is clear that the trucking sector has very important impacts on our economy, and yet the amount of research done within this sector is not much. Thus, to study and evaluate the need to have comprehensive models of this sector that consider the key incentives, constraints an extensive research is a must. This comprehensive modelling approach would allow us to consider a wide-range of optimised and regulated strategies so that the most effective ones are pursued. It would also allow us to identify and mitigate market failures in this sector by gaining an understanding of the how the current operations of the industry can be improved. However, the analysis on load body of truck presently does not exist, primarily because of limitations in the data that is publically available. This paper seeks to provide policy makers with better methodologies for modelling this sector; that make the most out of the scarce data that is available, so that the best policies can be pursued in this critically important sector of our economy. In the present work the aim is to optimize the load body mounted on the runner by minimizing the welding done during mounting of the load body on the chassis of the Heavy Commercial Vehicle (HCV) thereby reducing the weight to a certain extent and minimizing the cost due to welding, which should be efficient than the existing model and satisfy all the requirements according to Automotive Research Association of India. The optimized Finite Element Method (FEM) results will be compared with the existing on the basis of stress, deformation and weight. To evaluate the structural performance of the load body, it is necessary to determine stresses and deformations occurring at different sections. Also, once the test data is available, it can be used for correlating it with the FEA results and can be used for reliable analysis to develop the product. The methodology helps in moving towards virtual testing and to avoid/reduce physical testing of the sub-systems to adhere to the constraints on time and cost.

The main motivation for this project lies in the fact that the scope for technological improvements in HCV segment in India is very high. There has not been much research on improving the strength and stiffness of HCV load body. Major improvements provided to the HCV load body with appropriate optimization can prove beneficial and economical.

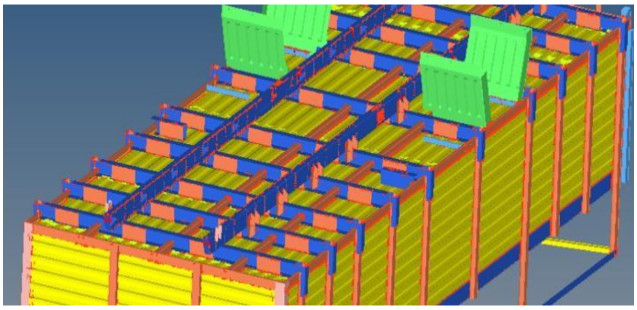
2. Methodology

The methodology used for the project has been specified in detail below:
1. Design - The load body carrier design
2. Model - 3D Modelling of the Load Body Carrier in CATIA V5R19
3. Meshing - Meshing using HyperMesh
4. Analysis - FE Analysis in OptiStruct
5. Results - Viewed in HyperView

The designing and modelling of load body was completed in CATIA software and the meshing, analysis and results were performed in Hypermesh software. In this section, a detailed overview about stitch welding will be explained. A lot of engineers and welders are confused about the difference between stitch welding and seam welding since they are often used interchangeably, but there is a definite difference between the two types of welds. In the welding fabrication world, it is important to distinguish the two because both welds have different strength, properties and are used for different purposes.Stitch welds generally have two numbers next to the weld call out to represent the length of the stitch weld and the pitch of the weld. The length is the length of the actual stitch weld. Whereas the pitch number is the distance between the centers of two stitch welds. Below are photos of each type of weld for better comparison.

Fig.1 Fillet Seam weld

Fig.2 Fillet Stitch weld



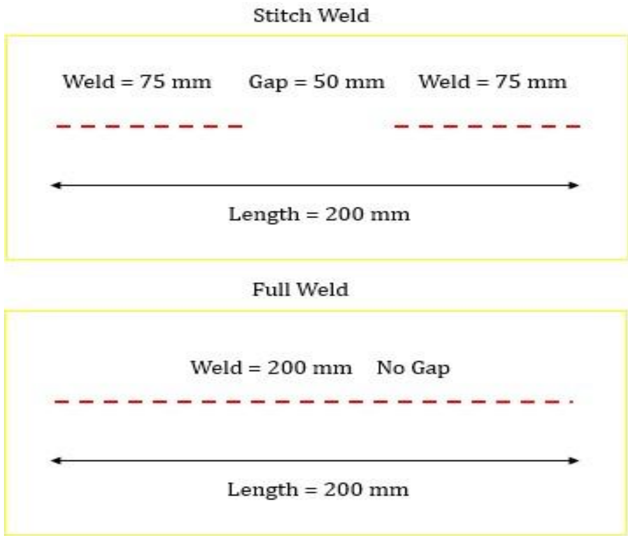


Fig.4 Conceptual representation of stitch weld

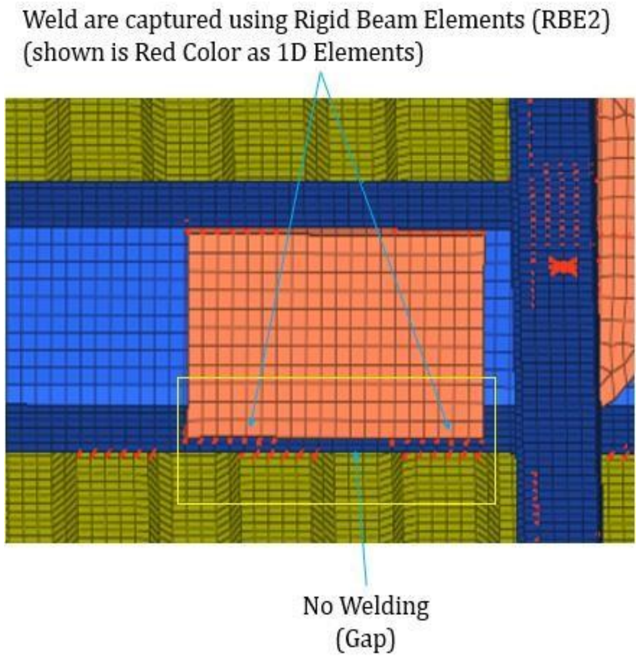


Fig.3 Stitch weld performed on load

A stitch weld is also known as an intermittent weld. Stitch welding is not a continuous weld across a joint, but a weld broken up by space gaps in between welds, which results in a “stitch” look. It certainly is not as robust and durable as a seam weld. It is used to prevent heat distortion and to also reduce the cost of welding if a long continuous weld is not necessary. The two common types of stitch welds are fillet weld and butt joint weld. A fillet weld is a triangular seam weld that joins two surfaces at right angles to each other, whereas butt welds are generally utilized in a lap joint, T-joint, or corner joint, which is most common amongst welding pipe or tubes to another surface where two pieces of metal are positioned coplanar and touching on one edge.

Fig.5 Magnified view of stitch weld

Above figures 3 and 5 depict stitch welding performed between C-channel (cross member) and its bracing using rigid elements (line elements).In this manner stitch welding is performed.

3. Design and Analysis of Load Body

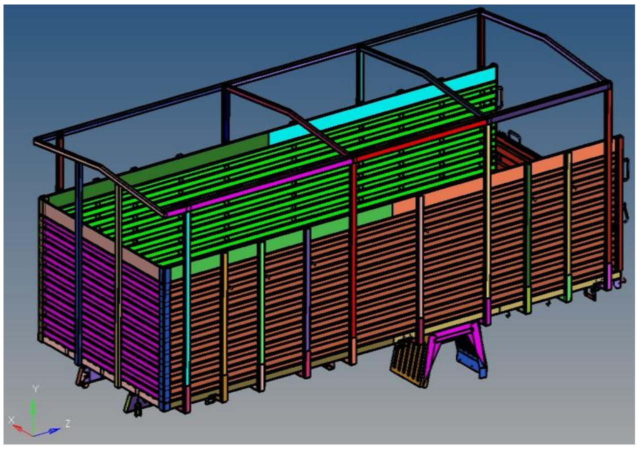


Fig.6 CAD model of load body

The specifications of the CAD model are mentioned below:
Length = 5880 mm
Width = 2280 mm
Deck Height = 1135 mm
Vehicle Type = HCV High Deck
Load Carrying Capacity = 11 Ton
Make = TATA Motors

It is important to establish a rational finite element model as it is crucial before the vehicle structural analysis. The model should sufficiently consider the balance between the structures and possess better precision. To facilitate the matter, a simplified finite element model is established accurately responding in the mechanical characteristics of the vehicle structure based on the premise of the vehicle structure and load characteristics.

In the process of developing the model of load body, a proper design method is employed. The conceptual design is first developed using sketches. Then, Computer Aided Design (CAD) software that is CATIA V5 R19 is utilized to help the process of the design. Once the three dimensional (3D) drawing was completed, a basic Finite Element Analysis was performed. The analysis and meshing work was done using Hypermesh software.. In Hypermesh, we have used 2D shell meshing on the model; in which maximum number of elements are first order quad and few are first order tria elements. The thickness of parts is too less because most of structure is sheet metal only. In such case, only 2D mesh is used in industry and that is the reason why we have performed 2D shell meshing.

It is important that the results of the analysis satisfy the technical requirement of the vehicle. In case the initial design of the structure does not satisfy the criteria for strength & load bearing capacity then the design will be changed and again reanalysis will be done. It is also essential to select appropriate material during designing of the load body. The material used is steel of FE410 grade and the details about the various properties of the material have been mentioned in the table below.

Table 1 Properties of material selected

S. No	Properties	Values
1	Yield Strength	250 MPa
2	Young's Modulus	2.1e5 MPa
3	Poisson's Ratio	0.3
4	Density	7.65 mg/m^3

4. Theoretical and Analytical Values

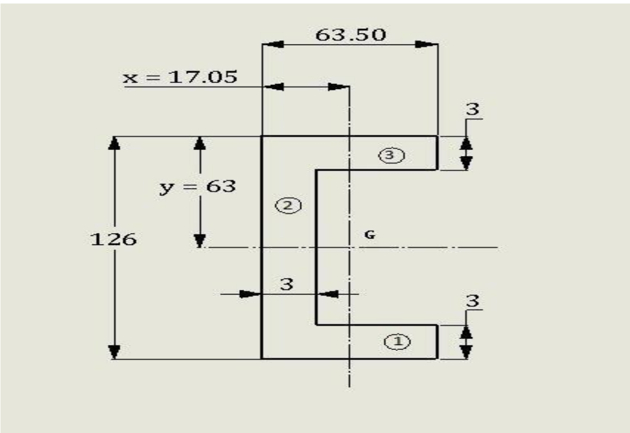


Fig.7 Symmetric C-section about x-axis

The location of y-axis for above C-section is given by,

$$\bar{x} = \frac{A_1x_1 + A_2x_2 + A_3x_3}{A_1 + A_2 + A_3} \quad (1)$$

$A_1 = 3 \times 36.5 = 190.5$ mm^4
$A_2 = 3 \times 120 = 360$mm^4 $A_3 = 3 \times 36.5 = 190.5$ mm^4 $x_1 = 63.5/2 = 37.5$ mm $x_2 = 3/2 = 1.5$ mm $x_3 = 63.5/2 = 37.5$ mm

Substituting the above values in eq. (1) we have,
$\bar{x}$=17.054 mm from L.H.S.
The given C-section is symmetric about x-axis
$\therefore \bar{y} = 63$ mm
Moment of Inertia for given C-section about x-axis,

$I_{xx} = I_{xx1} + I_{xx2} + I_{xx3}$(2)
IG = bd^3/12
Moment of inetria of rectangle 1 about horizontal xaxis of C-section is given by Parallel Axis Theorem.

$I_{xx1} = IG_1 + A_1h^2{}_1$ (3)
Where,
$h_1 = 63 - 1.5 = 61.5$ mm

$$IG_1 = \frac{63.5 \times 3_3}{12} = 142.875 \text{ mm}^4$$

Substituting above values in eq. (3), we have
$\therefore I_{xx1} = 720661.5$ mm^4
Similarly,

$I_{xx2} = IG2 + A2h22$

Where $h_2 = 0$ mm

$$IG^2 = \frac{3 \times 120^3}{12} = 432000 \text{ mm}^4$$

$\therefore I_{xx2}$ =432000 mm^4

$I_{xx3} = IG3 + A3h23$

$I_{xx3} = I_{xx1}$ (Symmetric about x-axis)

$\therefore I_{xx3} = 720661.5$mm^4
Substituting all the values in eq. (2)
$\therefore I_{xx}$= 1873323mm^4
Moment of Inertia for given C-section about y-axis,

$I_{yy} = I_{yy1} + I_{yy2} + I_{yy3}$(4)
IG = db^3/12
Moment of inetria of rectangle 1 about horizontal xaxis of C-section is given by Parallel Axis Theorem.

$I_{yy1} = IG_1 + A_1h^2{}_1$ (5)
Where,
$h_1 = (63.5/2) - 17.05 = 14.7$ mm

$$IG_1 = \frac{3 \times 63.5^3}{12} = 64011.96875 \text{ mm}^4$$

Substituting above values in eq. (5), we have
$\therefore I_{yy1}$ =105177.11 mm^4
Similarly,

$I_{yy2} = IG2 + A2h22$

Where $h_2 = 17.05 - 1.5 = 15.55$ mm

$$IG^2 = \frac{120 \times 3^3}{12} = 270 \text{ mm}^4$$

$\therefore I_{yy2}$ =87318.9 mm^4

$I_{yy3} = IG3 + A3h23$

$I_{yy3} = I_{yy1}$ (Symmetric about y-axis)

$\therefore I_{yy3} = 105177.11$mm^4
Substituting all the values in eq. (2)
$\therefore I_{yy}$= 297673.12mm^4

$I = I_{xx} + I_{yy}$= 2170996.12 mm^4 (6)

$$\delta_{max} = \frac{Pl^3}{48EI} \quad (7)$$

Total load applied is 11000 kilograms.
Thus total load applied in Newtons
= 11000×9.81 N = 107910 N.
Hence the amount of load acting on each C-section
= 107910/40 N
$\therefore P$= 2697.75 N
The length l =790 mm
Young's modulus E = 2.1×10^5 MPa

Substituting all the values in eq.(7), we have

δ_{max}= 0.0607 mm

Moreover,

$$\delta = \frac{\sigma l}{E} \quad (8)$$

Where,

$\sigma$is the stress induced.

Thus substituting all the determined values in eq.(8), we have $\sigma = 16.13$ MPa.

According to Finite Element Analysis(FEA), the maximum stress induced in C-section is 17 Mpa.

Thus it can be stated that the value calculated thereotically and analytically are almost similar.

5. Results and Discussion

5.1 Results under Gravity load

In the first case, we have considered the stresses and deformation that take place in body only under gravitational load. The results obtained have been specified in the figures below and they are well within the specified limits.

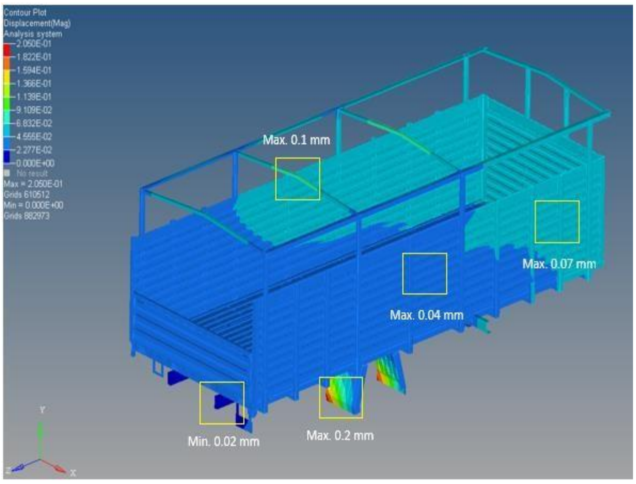


Fig.8 Deformations under gravity load

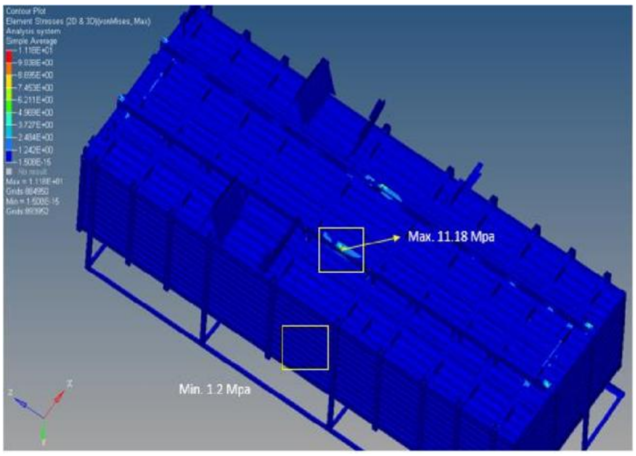


Fig.9 Stresses under gravity load

5.2Results under payload of 11 tonnes In the second case, we have considered the stresses and deformation that take place in body under payload of 11 tons. In this case, the maximum amount of stress that occurred in the load body is 71 MPa and in full weld section is 17 MPa. During theoretical calculations the value was 16.13 MPa, thus it can be stated that there is similarity among the theoretical and analytical values. Existing Design is very much safe for loading under 11 tons. The allowable stress limit (Yield Strength) for the material is 250 MPa and the max stress in this case is only 71 MPa which is far less compared to yield strength. In case of full weld section the maximum stress is only 17 MPa so it's safe as well.

Thus there is scope for design optimization over here. One can optimize its geometry, Section Size, Thickness etc. for future scope. We are focusing on weld optimization in this paper. Hence next version of analysis is with stitch weld instead of full or continuous weld.

Fig.12 Stresses in full weld section

5.3Results under optimized stitch weld In the third case, we have considered the stresses and deformation that take place in the optimized load body under payload of 11 tons. In this case, the maximum amount of stress that occurred in the load body is 88 MPa and in stitch weld section is 51 MPa. Existing design with

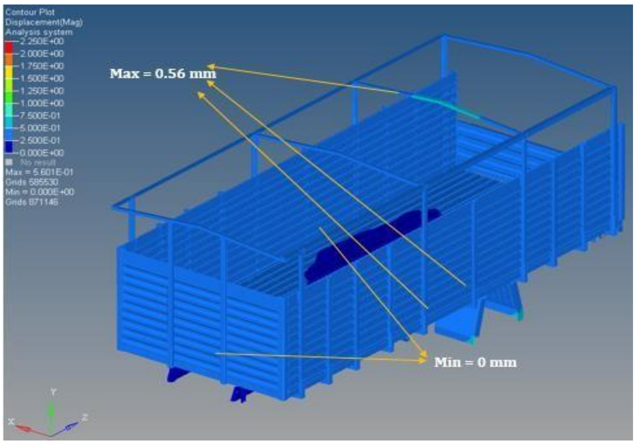


Fig.10 Deformations under payload

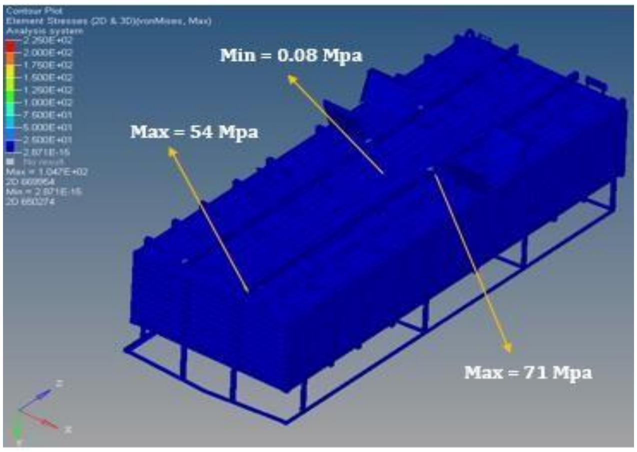
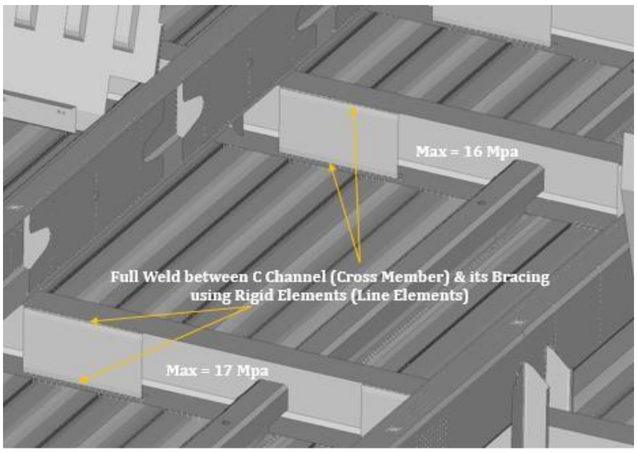


Fig.11 Stresses under payload



full weld was seen very much safe for loading as per earlier results. Similarly these results for stitch welds are also safe &within limit of safety. Allowable stress limit (Yield Strength) is 250 MPa and the maximum stresses are less in comparison. Thus, it can be stated is that the design optimization.

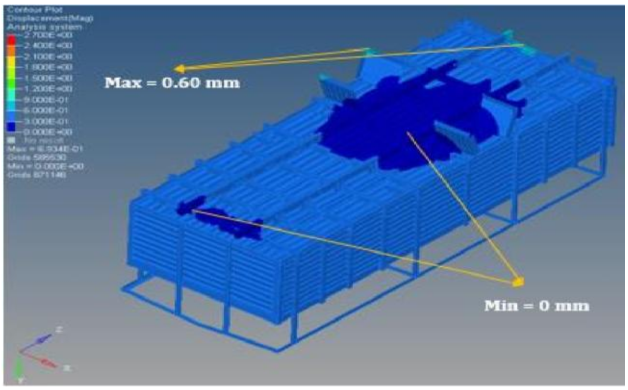


Fig.12 Deformations in optimized load body

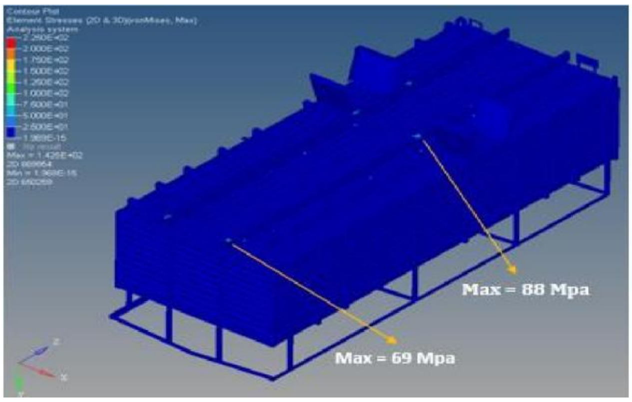
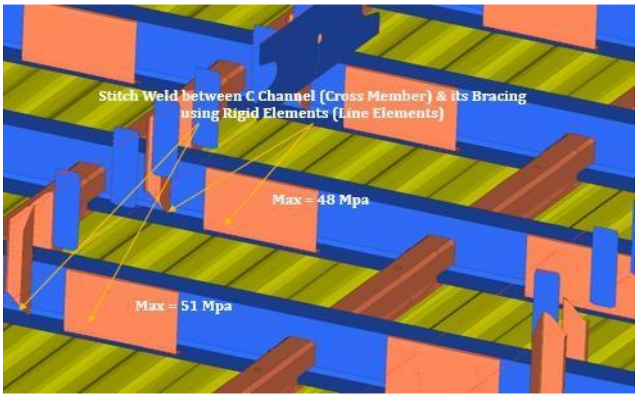


Fig.13 Stresses in optimized load body



safe after its

Fig.14 Stresses in Stitch weld section of optimized load body

Conclusions

1) From the results, it is clear that stitch welding can be recommended for load body since the stresses generated are within permissible limits.
2) Stitch welding can reduce the labour costs as compared to full welding.
3) Optimizing the design as specified above once can expect a marginal reduction in weight.
4) The amount of lead time will be reduced when stitch welding is preferred.

References

Sebastian E. Guerrero, Samer M. Madanat, Robert C. Leachman, (2013), The Trucking Sector Optimization Model: A tool for predicting carrier and shipper responses to policies aiming to reduce GHG emissions, *Elsevier, Transportation Research Part E 59*,pp. 85-107.

Mingming Wang, Tengfei L, Xin Li, Cheng Liu, and Huixia Liu, (2014), Body Structure Static-Dynamic Analysis and Optimization of a Commercial Vehicle, *Key engineering materials*,62,pp. 400-406.

Dakin J, Heyes P and Fermer Metal, (2001) Analytical Methods for durability in Automotive Industry-Engineering process, *SAE Interntional*, pp. 21-32.

Yi Li, Bo Son and Wen Zhao, (2011), A New Method of Reliability Analysis on Strength and Stiffness for Frame Structure, Advanced materials research, 165, pp. 3411-3415.

Muhammad Zahir Hassan, Hendrie John,(2015), Design of Chassis Frame for All-Terrain Vehicle for Educational Purposes*, Applied Mechanics and Materials*,695, pp.742-745.