International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 10 Issue: 04 | Apr 2023 www.irjet.net p-ISSN: 2395-0072

Strength Improvement of Bus Body Structure with Design Modification

Praveen Kokitkar[1], Adarsh Gatade[2], Rohit Lokolkar[3], Nitin Chopade[4], Rohit Mudekar [5], Samarth Kashid[6]

[1]Assistant Professor of Mechanical Department, Dr. A. D. Shinde Collage of Engineering, Gadhinglaj-416502, Maharashtra, India
[2,3,4,5,6] Student of Mechanical Department, Dr. A. D. Shinde Collage of Engineering, Gadhinglaj-416502, Maharashtra, India

---***---

Abstract - *In our society mobility conditions are adding drastically with growing population and contemporaneously environmental perceptivity is also adding. Thus it's making a big challenge for transport pots of government and private drivers to manage vehicles and business policy makers to handle them. Accordingly, the light and heavy passenger vehicle manufacturers as well as vehicle body builders have to borrow new request conditions along with minimal introductory safety of the passenger.*

The design includes detailed design, development and analysis of the body structure of machine. According to AIS law, performance parameters have to be followed by the machine body builders to insure needed strength. To insure strength of the machine body, all the machine body builders have to assay their design in any one of the FEA(Finite Element Analysis) tool. With the help of FEA tool bone can gain accurate results of strength of design in lower time. As notified in Rule number 126 of the CMVR 1989, all machine body builders have to use anyone of the FEA tools by themselves or with authorised test agencies to corroborate the strength.

Key Words: **Bus Body Structure, FEA, Strength Improvement, Vehicle Body Builders**

1. INTRODUCTION

All the OE vehicle manufacture sells their vehicles in drive away chassis forms, so that customer can get vehicle body buildings don as per his requirements and choice with local body builders with low cost. These local body builders use low quality parts and materials in a poorly designed body which leads to accident some time later. These body structures are not safe since designs are not optimal, for example these designs provide more heat transfer, noise, and vibrations inside the cabin.

All the automobile body builders have a big demand for compact body and can accommodate more number of passengers with good safety of all the passengers. Therefore these new body designs must have high strength to withstand more load of increasing passengers. The strength of body is analysed and verified through the Finite Element Analysis (FEA) software and life security of the users can be guaranteed.

1.1 Problem Description

In MSRTC workshop nearly 2000 number of bus are manufactured per year. The periodic conditioning of these buses will be done after every three years or after running every three lack kilometres. In most of the cases it was observed that failure occurs at joints at clits or pillars, as we can see in figure 1 & 2. So there will be requirement for the replacement of the failed members to recondition. This replacement of components is very time consuming and tedious job. These buses are designed to survive at least a minimum of ten lack kilometres. Therefore high strength designs, materials and components must be used.

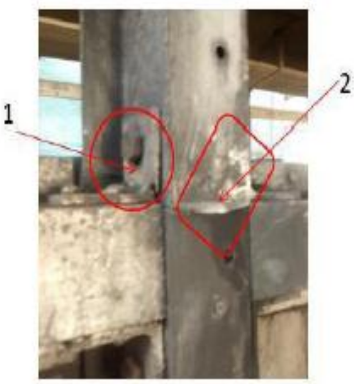

Fig -1: Clits break at joints

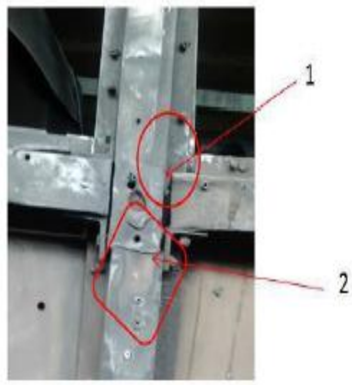

Fig -2: Pillars break near joints

1.2 Objectives of the project

Demand for public transport is increasing day by day. To ensure strength of the newly innovatively designed component, it has to be tested and analysed. These are the objectives of the project

1. To carry out the detailed the study of the structure of bus body as per AIS052 standard.
2. To study all the bus body structure theory.

3. Failure modes are studied and cause-effect diagram is prepared.
4. To prepare Cause-effect diagram. To do the study of failure modes by components structural analysis and to find critical region for failure using suitable analysis software.
5. To provide solution to avoid failure of structure of bus.
6. To carry out analysis of modified structure using analysis software.
7. To validate the simulation results of modified structure of bus body by comparing the results with simulation results of existing structure of bus body.

2. Dimensional specifications for a bus body

Specifications used in MSRTC- 2x2 parivartan bus are as follows. Below figure shows the dimensions used in MSRTC-2x2 parivartan bus with the standards given in AIS_052

Table -1: MSRTC- 2x2 Parivartan dimension specification

Components		Dimension (mm)	
		AIS-O52	Actual
Max seating capacity			44+11+2
1	Overall widthmax	2600	2570
2	Overall lengthmax	12000	10370
3	Overall heightmax	3800	3095
4	Wheel Base	5895	5895
5	Rear Over hang	3537	3285
6	Front Over hang	1187	1187
7	Min. no of service doors	1	1
8	Min width of Service Door(front single Door)	650	690
9	Height of service Door	1650	2050
10	Min width of windows	550	558
11	Guard rails (Min 2)	2	2
12	Emergency Exits	2	2
13	Gung way min width	350	430

3. Layout of Framework of a bus body

Layout of a bus body is shown in the below figure with dimensions. Material used for body is aluminium. The framework is done by using M8 bolts with riveted joints. Fig shows the assembly of bus body according to dimensions.

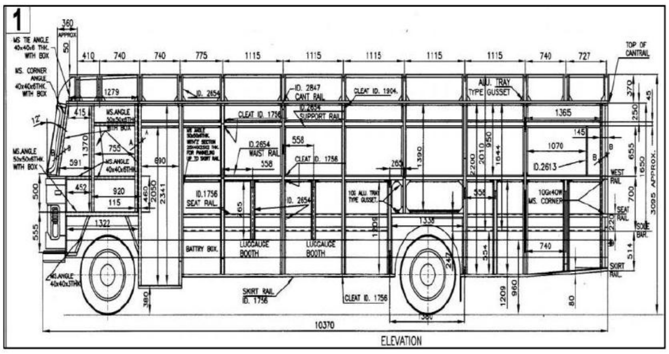

Fig.3 Left side view of the bus body frame

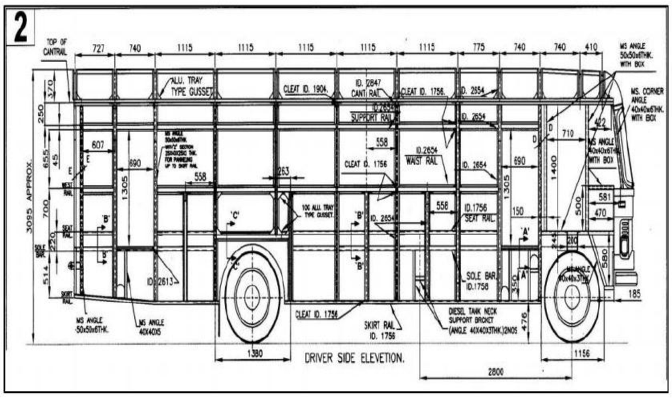

Fig.4 Right side view of the bus body frame

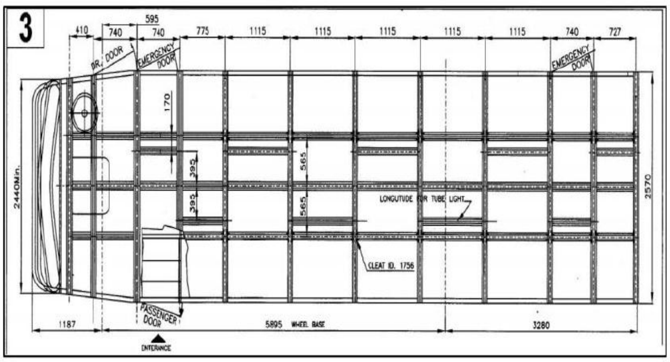

Fig. 5 Top (roof) frame view of bus body

4. Finite Element Analysis

4.1 Static Analysis

Boundary conditions are used to restrict the body with constraints and different types of loads. In this problem bottom structure is attached to the chassis of the vehicle that is the long members of the bottom structures are connected. Therefore the long members of the bottom frame are constrained with all degrees of freedom. Also the load of 5Kg in X direction, 5Kg in Y direction and 10Kg in Z direction is applied. The loads applied can be seen in figure

Therefore the forces acting are

X direction = 5 x 9810 =49050 N

Y direction = 5 x 9810 = 49050 N

Z direction = 10 x 9810 = 98100 N

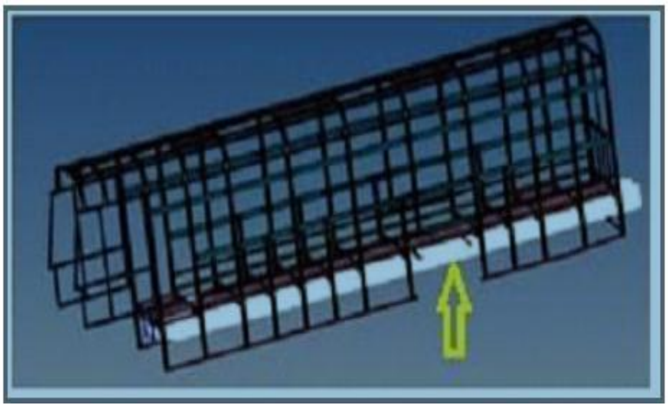

Fig. 6 Applying boundary conditions

4.1.1 Static Analysis Result

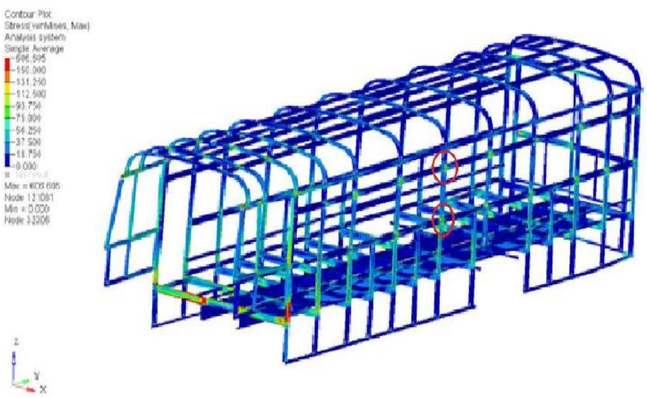

Fig. 7 Stress distribution

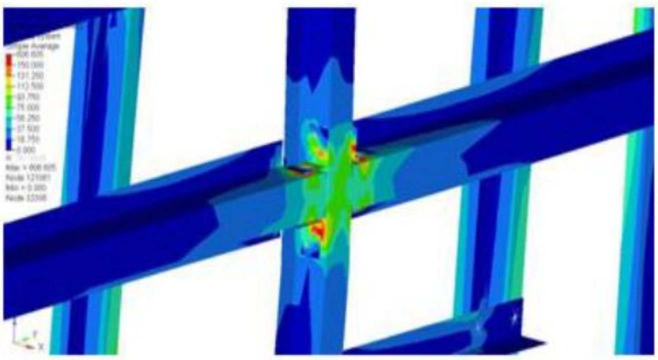

Fig. 8 Failure part

Stresses acting on the body structure can be seen in counter plot. The result shows the stress distribution in the body structure which can be seen in figure 6.3. In the figure it can be also seen that the stress concentration is more at the clits. This is the part which is prone to failure and which can fail at the earliest during accidents.

It can be seen in figure 6.4 that this is the critical area which will fail on repeated loads and timely maintenance is required. Therefore this part of the structure needs to be modified and strengthened to improve the life of the body structure.

4.2 Dynamic Analysis

Dynamic type of analysis is divided into different types, they are as follows

1. Modal

2. Frequency response

3. Transient response

4. Random vibration

4.2.1 Modal Analysis

This analysis is carried out to find the systems natural frequency.

Table No. 2 Natural Frequency

Mode Shape	Natural Frequency (Hz)
1st	6.110
2nd	7.684
3rd	8.310
4th	9.160
5th	11.08
6th	12.73
7th	15.41
8th	18.06
9th	22.80
10th	25.12
11th	26.80
12th	28.30
13th	32.38
14th	42.41
15th	52.00
16th	55.20
17th	58.12
18th	60.00

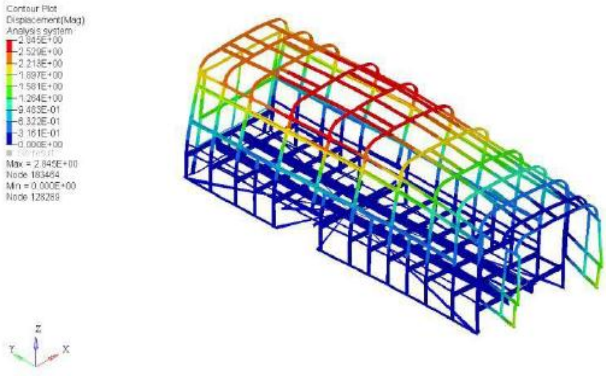

Fig. 9 Mode at 6.110Hz (Lateral bending)

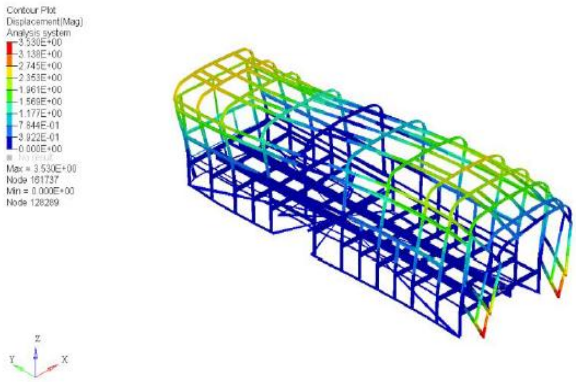

Fig. 10 Mode at 7.684Hz (Lateral bending)

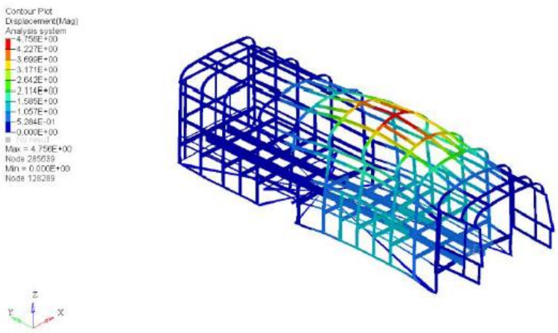

Fig. 11 Mode at 11.08Hz (Roof bending)

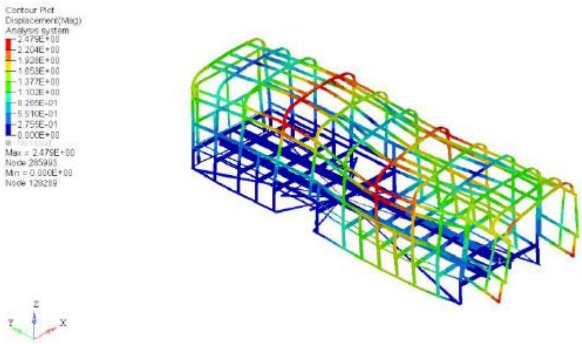

Fig. 12 Mode at 12.73Hz (Twisting)

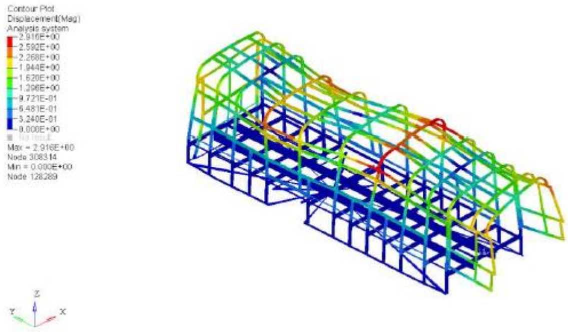

Fig. 13 Mode at 15.41Hz (Shearing)

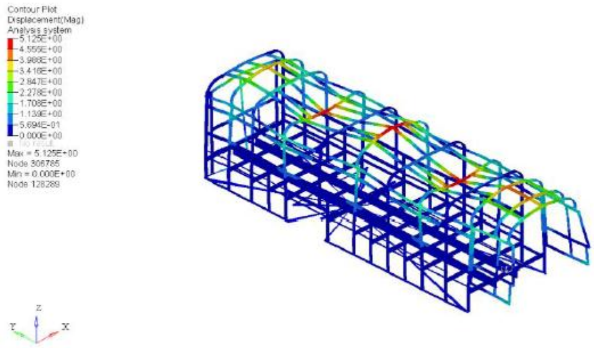

Fig. 14 Mode at 18.06Hz (Roof bending)

4.2.2 Frequency/response type analysis

The figures below show the result for frequency response type of analysis. Figure 6.13 shows value of maximum displacement of 3.117 m. Figure 6.14 shows the result for equivalent type of stress. It was observed that maximum stress was 63.244 MPa and the maximum stress at clits was 49.447 MPa.

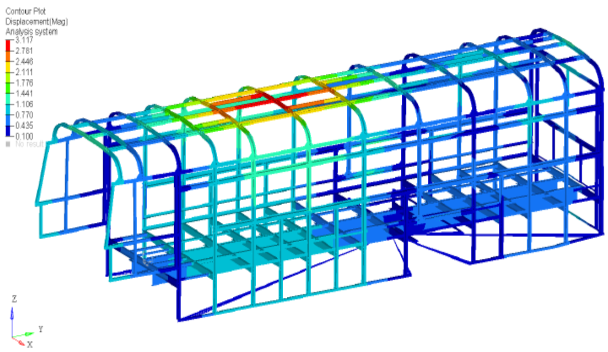

Fig. 16 Displacement

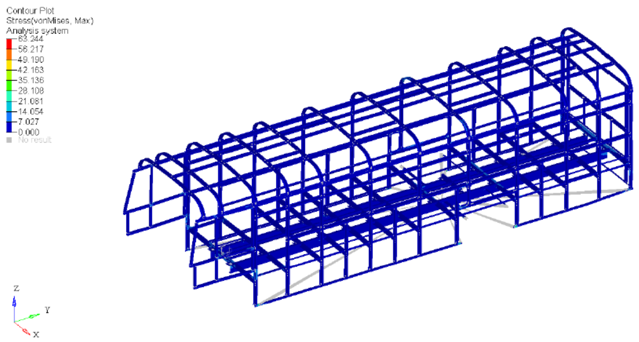

Fig. 17 Stress distribution

5. Modification Required for Clit

Due to the problem faced by the present design, the clit was required a new design to improve the strength. Therefore clit was redesigned with the dimensions 76.2 x 50.8 x 6 mm. Material selected is same as that of previous clit. Figure 7.2 shows the new design of the clit.

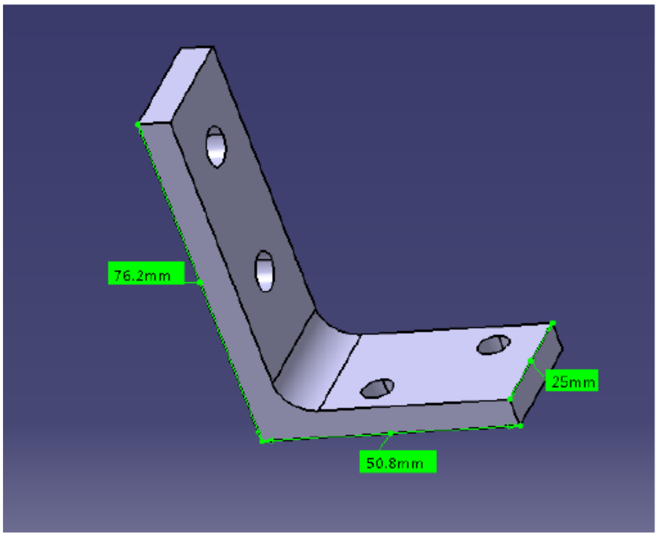

Fig.18 Modified design of clit

5.1 Analysis results

After redesigning the clit in the CAD software, the clit was replaced with the new model and again meshed and analysed for all the conditioned which was followed earlier. For frequency response analysis it was found that maximum displacement was 1.603m (figure 6.3) and maximum stress of 35.71 MPa (figure).

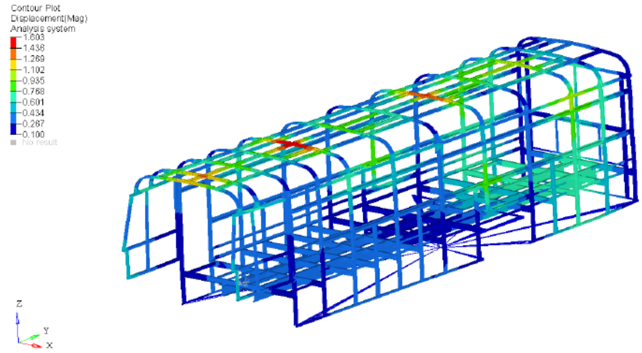

Fig. 19 Maximum displacement

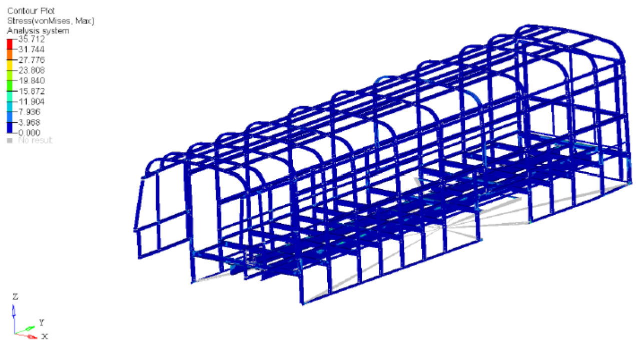

Fig. 20 Stress distribution

6. Conclusion

In the project the MSRTC bus body structure was designed and analysed because some parts were giving lot of maintenance issues. This problem was solved by redesigning the clit part of the bus body. Both old and new designs were analysed and compared. The below table number 8.1 shows the comparison of new and old design analysis.

Table 8.1 Conclusion comparison

	Existing Structure	Modified Structure
Displacement	3.117	1.603
Von-Mises Stress	63.244	35.712
Stress in Clint	49.4	26.02

By seeing above table we can understand that with the modified design the structure of bus body can withstand more load and lasts longer.

ACKNOWLEDGEMENT

It gives us an immense pleasure to write an acknowledgement to this Paper, a contribution of all people who helped us realize it. We take this opportunity to express our respectful regards to our beloved **Principal Prof. K. S.**

Joshi for motivate us to publish this paper. Also we express our deep sense of gratitude and appreciation to our beloved **Project Guide Prof. Praveen Kokitakar** for this enthusiastic inspiration and amicable in all phases of our Paper.

REFERENCES

A. Magnus G.H. Cruz, AlexandreViecelli, "A methodology for replacement of conventional steel by micro alloyed steel in bus tubular structures", Materials and Design 29, Elsevier,pp.539-545, 2008.

B. J. Karliński, M. Ptak, P. Działak, E. Rusiński, "Strength analysis of bus superstructure according to Regulation No. 66 of UN/ECE", Archives of civil and Mechanical Engineering 14, Elsevier, pp.342–353, 2014.

C. A. Gauchía, E. Olmeda, M. J. L. Boada, B. L. Boada and V. Díaz, "Methodology for bus structure torsion stiffness and natural vibration frequency prediction based on a dimensional analysis approach", International Journal of Automotive Technology, Vol. 15, No.3, pp. 451–461, 2014.

D. G. Zhang, X. Zhang, and B. Liu, "The Study of Bus Superstructure Strength Based on Rollover Test Using Body Sections", Advances in MSEC Vol. 2, AISC 129, Springer, pp.315–322, 2011.

E. H. S. Kim, Y. S. Hwang, H. S. Yoon, "Dynamic Stress Analysis of a Bus Systems", Annual Hyundai Motor Company Conference, Korea, 2009.

F. SachinThorat, G.Amba Prasad Rao, "Computational analysis of intercity bus Withimproved aesthetics and aerodynamic performanceonIndian roads", International Journal of Advanced Engineering Technology, E-ISSN, pp.0976- 3945,2011.

G. S. Butdee, F. Vignat, "TRIZ method for light weight bus bodystructure design",Achievements in Material and Manufacturing Engineering, Vol 31, Issue 2, 2008.

H. Prasannapriya. Chinta ,Dr.L.V.VenugopalRao, "A New Design and Analysis of Bus Body Structure", IOSR Journal of Mechanical and Civil Engineering, e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, Issue 5 VolI,pp39-47, 2014.

I. Automotive Industry Standard-052, “Code of Practice for Bus Body Design and Approval” IS 733-1983 “Specification for wrought Aluminum and Aluminum alloy bars, rods and Sections”

J. Nitin Gokhale, “Practical Finite Element Analysis”, 2007