HAL
open science

Finite Element Structural Analysis of a Pit Viper 235 Main Frame

Anand Sn, Manohar Hs, Kumar Thanaiah

To cite this version:

Anand Sn, Manohar Hs, Kumar Thanaiah. Finite Element Structural Analysis of a Pit Viper 235 Main Frame. Journal of Scholastic Engineering Science and Management, 2021, 1 (3), pp.19-31. hal-03284127

HAL Id: hal-03284127

https://hal.science/hal-03284127v1

Submitted on 23 Jul 2021

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

JSESM

Journal of Scholastic Engineering Science and Management

June 2021, Volume 1, Issue 3, pp: 19-31

"Finite Element Structural Analysis of a Pit Viper 235 Main Frame"

[1]Dr. Anand Sn, [2]Dr. Manohar Hs, [3]Kumar Thanaiah

[1]Associate professor, School of Mechanical Engineering, Debre Bethan University, Debre BethanCity, Ethiopia

[2]Professor and Head, Department of Mechanical Engineering, Dayananda Sagar Academy of Technology and Management, Bangalore, Karnataka, India

[3]Lecturer, School of Mechanical Engineering, Bule Hora University, Ethiopia

Email: [1]snagowda1974@gmail.com [2]hodmech@dsatm.edu.in [3]kumsmechs@gmail.com

ABSTRACT

The Pit Viper series 235 (PV-235) blast hole drill offers new technological innovations for many applications including hard rock mining for metals and overburden coal mining. This paper is centered on an ongoing structural analysis of PV-235 Mainframe. Systematic approach has been followed in this project using ANSYS FEA software. The methodology of this project is to carry out structural analysis of the PV-235 mainframe which is subjected to two different types of loading corresponding to the respective boundary condition. From the structural analysis for the given loading and boundary condition it is evident that the maximum stress on the PV-235 Mainframe is below the Yield stress of the material, so the design is safe under the given loading. Reaction Forces applied in the ANSYS tool also matches well with the given input which is done as a sanity check for this analysis.

Keywords: *Finite Element Analysis, PV (Pit Viper)-235 Mainframe Base, ANSYS Tool*

1. OBJECTIVES AND SCOPE OF THE WORK

Objective of this project is to evaluate the stress levels in the Pit Viper - 235 Mainframe by performing Pre-load analysis which involves pretension analysis of bolt to simulate the bolted or assembled condition of bolt & studs and to carry nonlinear analysis of PV-235 Mainframe assembly for the self-weight of the mainframe with all the weights attached to it and the Tower inHorizontal position.

Below are the some of the scope of the current thesis:

- FE model preparation of PV-235 Mainframe using HYPERMESH.
- Applying Loading and boundary conditions using ANSYS.
- Pretension analysis of bolts and self-weight analysis of the mainframe using ANSYS.
- Result's interpretation using ANSYS.

2. FINITE ELEMENT ANALYSIS PROCESS

Outline of Finite Element Analysis Process is given in Figure.1.0 Finite Element Analysis process can be put into two different objectives. First one is design analysis stage and second one is design changes based on the analysis carried out.

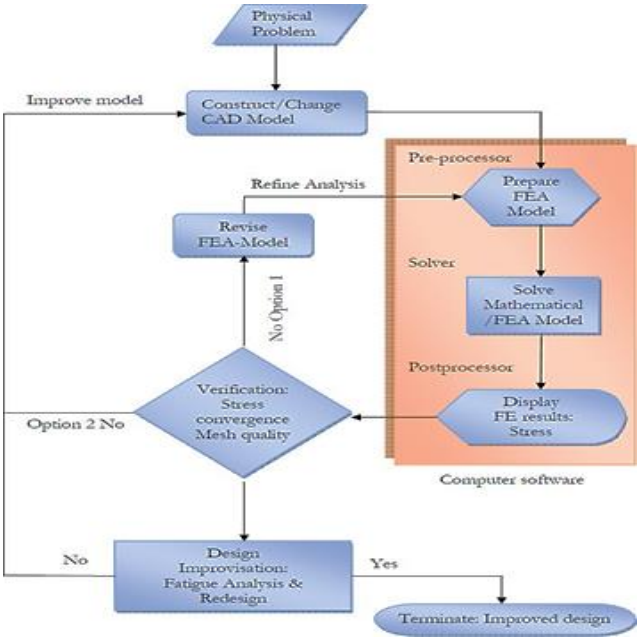


Fig.1.0: Outline of Finite Element Analysis Process

2.1. Physical Model of the PV235 Mainframe

The PV235 Mainframe is a completely bolted or used studs for joining cross member as represented in Figure 1.1. Complete Mainframe is made of ASTM A36 SS. The mechanical properties of the ASTM A36 SS are given in the Table 1.

Table 1: Mechanical properties of ASTM A36

Sl.No	Material	Yield strength (PSI)	Modulus of elasticity (PSI)	Poisson's ratio	Density (lb/inch^3)
1	ASTM A36	36300	2.9 x 10^7	0.3	0.283
Density of the Mainframe is calculated to match the mainframe self-weight of 21928 lbf.Density = 9.15e-4 Kg/inch^3					

3. FINITE ELEMENT MODELING DETAILS

Figure 1.1 give the FE Model details of PV235 Mainframe along with the component considered for the analysis. Here most of the components and meshed with solid or shellelements based on the requirement of the analysis. Bolting region is completely meshed with solid elements as shown in Figure 1.2.

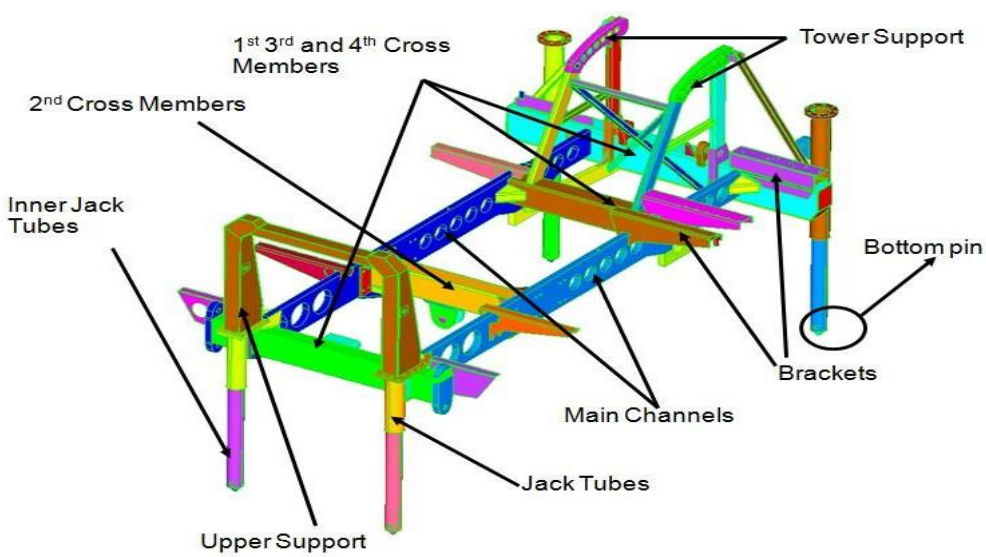


Fig 1.1: FE Model of PV235 Mainframe with component description and modeling details.

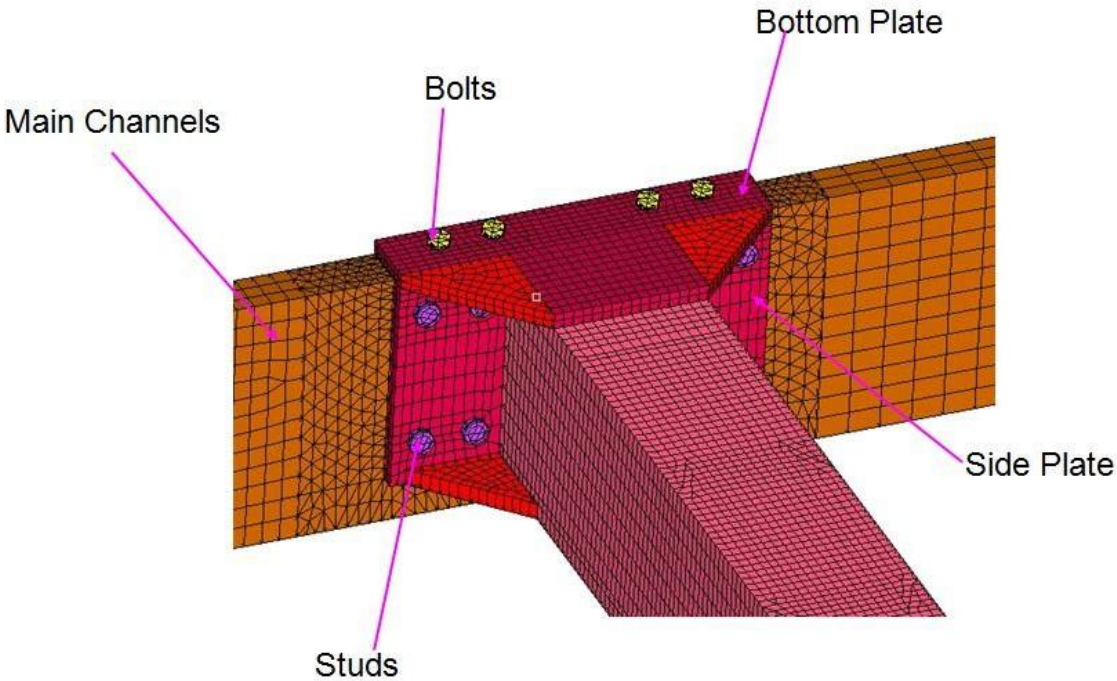


Fig.1.2: Bolting Details in the 2nd Cross member of PV-235 Mainframe

Connectivity used for PV235 Mainframe:

- Node to Node connectivity between Main Channels and Bolts.
- Standard contact between Main Channels and Bottom Plate.
- Standard contact between Main Channels and Side Plate.
- Bonded Contact between Main Channels and Stud Head.
- Bonded Contact between Bottom Plate and Bolt Head.

3.1. Elements used in Finite Element Analysis of PV235 Mainframe

Below are some of the elements used for this analysis and description of which can be founding ANSYS user manual

- SHELL181, 4-Node Structural Shell.
- SOLID185, 3-D 8-Node Structural Solid.
- PRETS179, Pretension element.
- CONTA174, 3-Dimensional 8-Noded Surface-to-Surface.

- TARGE170, 3-Dimensional Target Segment.
- MASS21, Structural Mass.

3.2. PV-235 Mainframe Boundary Conditions

In this theses Load case 1 is analyzed and reported, and below are the boundary conditions applied for the mainframe. Figure 1.3 gives the overall boundary conditions applied for the PV234 mainframe whereas the end of jack is constrained in all the translational direction meaning in all the X, Y and Z direction and allowed to have rotation degrees of freedom. Figure 1.4 gives the Constraints at jack, here the constraints is applied only on the 1.125" diameter (surface is flat).

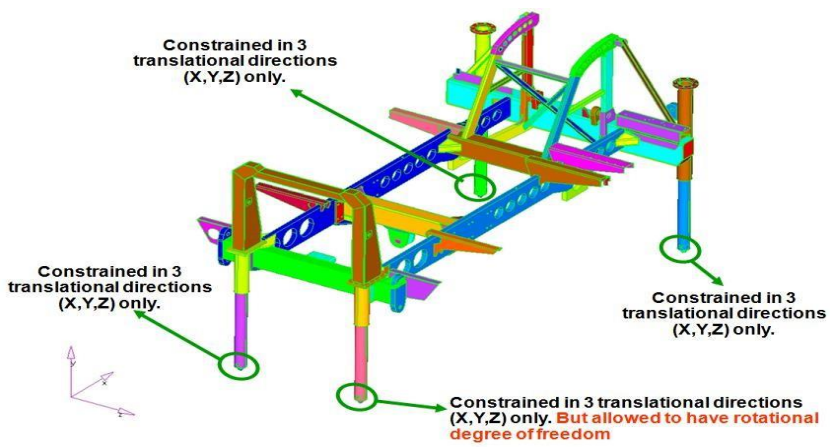


Fig.1.3: Boundary Conditions for PV235 Mainframe

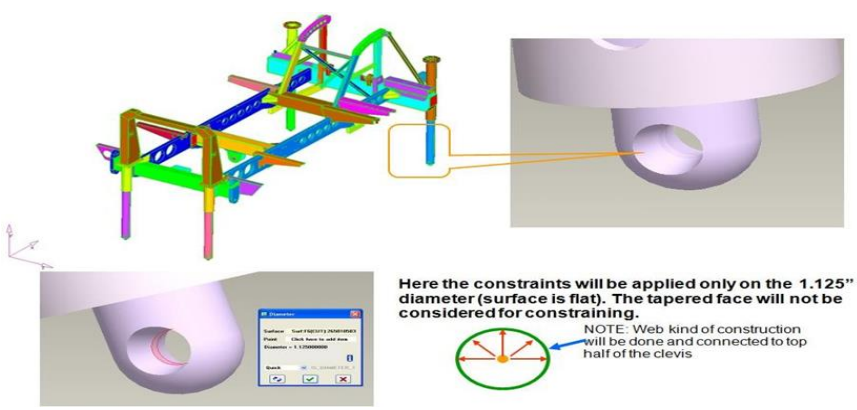


Fig.1.4: Constraints to jack for PV235 Mainframe

3.3. Loads on Mainframe Base and other loads

Complete loading summary for the PV235 Mainframe is given in the below table 2

Table 2. Loading summary for the PV235 Mainframe

SL No	Description	Weight (inlbf)	Weight inN	Comments
1	Rob Weight	8000	35584	Applied as reaction on tower support
2	Tower weight	23581	104888	
3	Lube	450	2002	On bolt holes
4	DHD	1200	5338	
5	Dust collector	2500	11120	
6	Cooler1	1500	6672	At Mainframe base and Mainframe mounting location
7	Cooler2	1500	6672	
8	Pump	1000	4448	
9	Engine	6500	28912	
10	Airend	2500	11120	
11	Mainframe base	1944	8645	
12	Receiver	1000	4448	
13	Fuel tank	5700	25354	
14	Water tank (Belly)	7520	33449	
15	Water tank (Deck mounted)	4400	19571	
16	Cab Weight	4500	20016	
17	Tracks weight total	28320	125967	
Total Weight		102115	454206	
Total rig Working weight		138000	613824	
Mainframe self-weight		21928	97536	
Non attributed mass		**13957**	**62082**	

4. RESULTS AND VALIDATION

4.1. Results of Load step 2: 1G Analysis Results

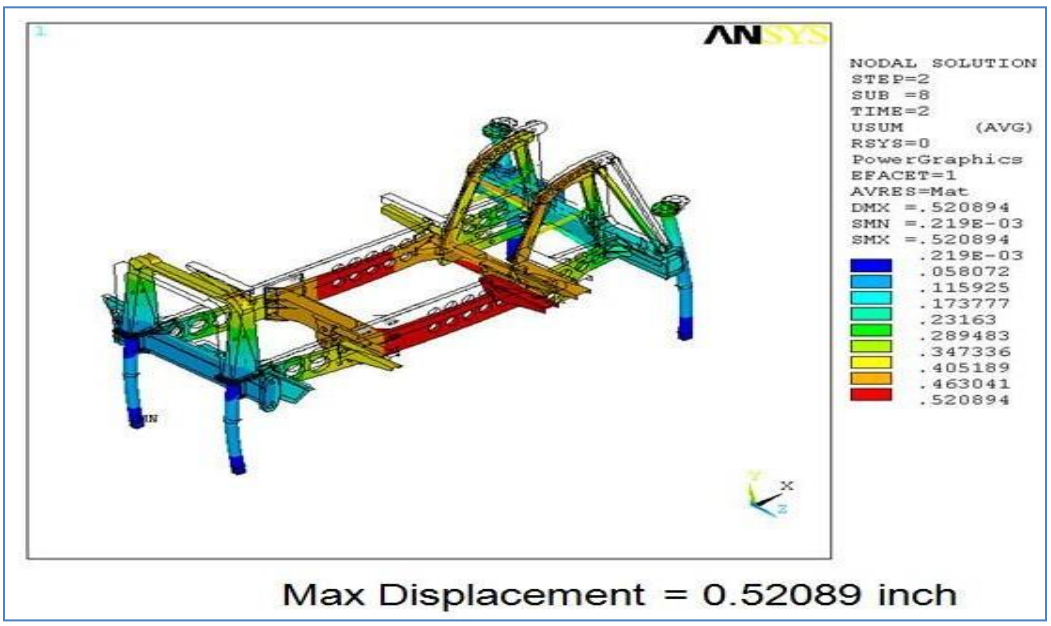


Fig.1.5: Overall displacement the PV235 Mainframe for Load case

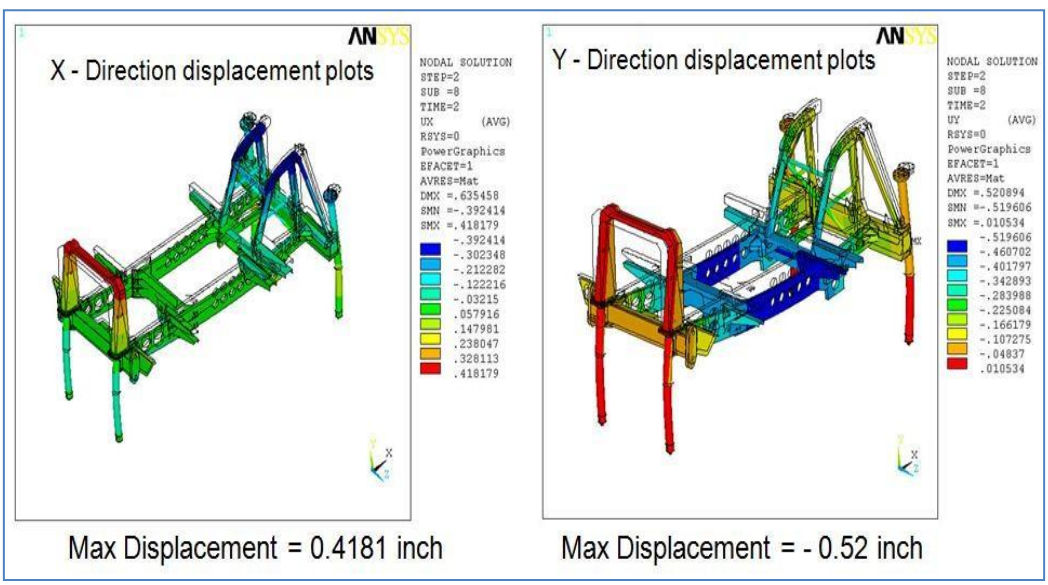


Fig.1.6: X and Y direction displacement the PV235 Mainframe

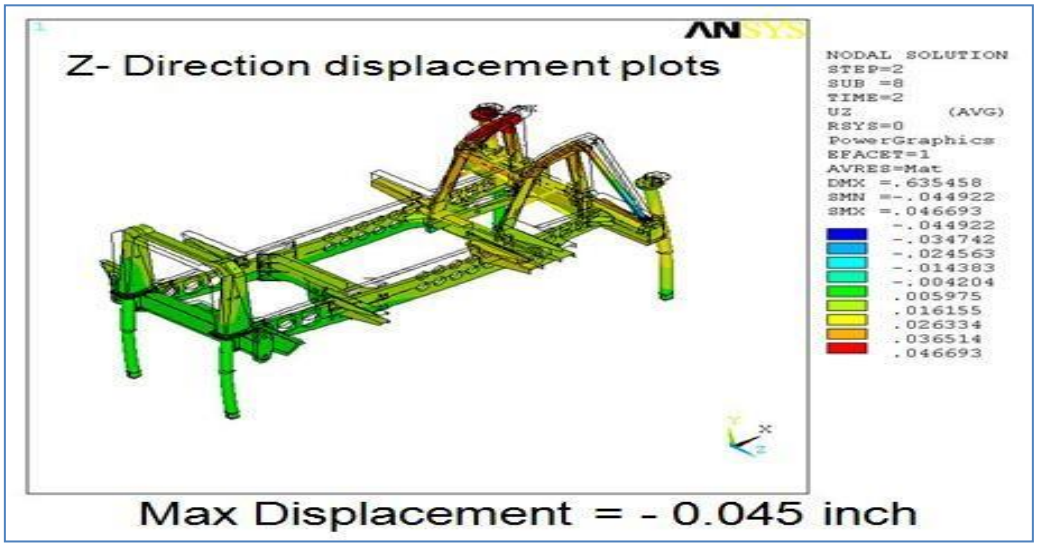


Fig.1.7: Z direction displacement the PV235 Mainframe

Fig.1.8: Von Mises stress Plot for PV235 Mainframe

Fig 1.9: First Principal Stress Plot for PV235 Mainframe

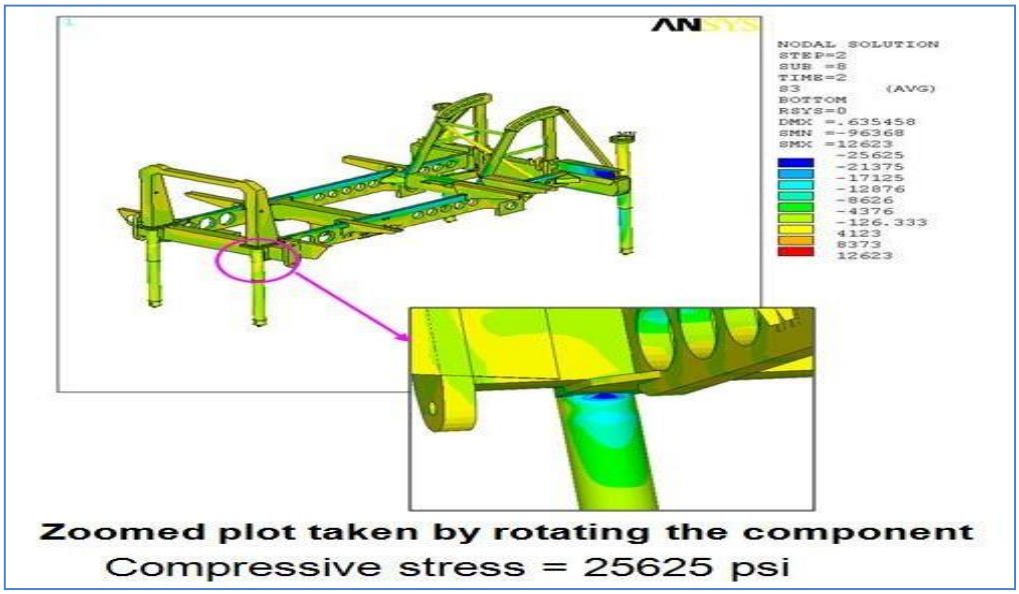


Fig.2.0: Third Principal Stress Plot for PV235 Mainframe

4.2.Results tabulation

Table 3 and 4 gives the results in tabular form for the Load step 1 and Load step 2 respectively.

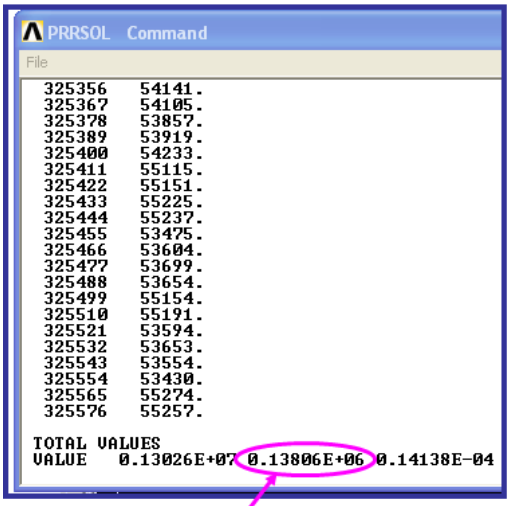
Table 3: Load step 1: Only bolt tightening load

SL No	Component	Stress in PSI		
		Von Mises	First principal	Third principal
1	Bolts	74116	82836	31176
2	Studs	72056	73212	39182
3	Bottom plate	35108	7104	35739
4	side plate	31045	12364	32261
5	Main channel	29152	15269	29173

Table 4: Load step 2: 1G Analysis Results

SL No	Component	Stress in PSI		
		Von Mises	First principal	Third principal
1	Main channel	24874	26587	19709
2	1st Cross member	11582	9233	11850
3	2nd Cross member	5036	5370	2834
4	3rd Cross member	18801	10627	20194
5	4th Cross member	22752	22993	15685
6	Rear Jack Tubes	22146	15775	25625
7	Front Jack Tubes	17890	16677	20556
8	Inner Jack Tubes	19883	20146	19923
9	Tower Supports	16629	12210	17783
10	Cross Angle	8340		
11	Support Brackets	26338		
12	Cab Bracket	8382		
13	Yoke Bracket	4017		
14	Gussets	6292		
15	Vertical Support	691		

4.3. Validation of Structural analyses through Reaction Forces calculation



Sl No	Description	Weight in lbf	Weight in N
1	Rob Weight	8000	35584
2	Tower weight	23581	104888
3	Lube	450	2002
4	DHD	1200	5338
5	Dust collector	2500	11120
6	Cooler1	1500	6672
7	Cooler2	1500	6672
8	Pump	1000	4448
9	Engine	6500	28912
10	Airend	2500	11120
11	Mainframe base	1944	8645
12	Receiver	1000	4448
13	Fuel tank	5700	25354
14	Water tank (Belly)	7520	33449
15	Water tank (Deck mounted)	4400	19571
16	Cab Weight	4500	20016
17	Tracks weight total	28320	125967
Total Weight		102115	454206
Total rig Working weight		138000	613824
Mainframe self weight		21928	97536
Non attributed mass		**13957**	**62082**

Reaction Force from analysis

Total Rig Working Weight = 138000 lbs

Fig.2.1: Reaction force from ANSYS Vs the Calculations

As confirmed from the about Figure 2.1. Reaction forces from the structural analysis matches well with the total rig working weight calculated summing all the working weights on the PV235 Mainframe. This confirms that the load applied is completely transferred to the Mainframe members and the stresses on the members are due the applied load.

CONCLUSIONS

Structural analysis for the Pit Viper - 235 Mainframe is performed for two cases first is the Pre-load analysis which involves pretension analysis of bolt to simulate the bolted or assembled condition of bolt & studs and to carry nonlinear analysis of PV-235 Mainframe assembly for the self-weight of the mainframe with all the weights attached to it and the Tower in Horizontal position. Based on the results of the structural analysis, below conclusions can be drawn.

- Stresses in the Bolt and studs due to pretension load are within the yield of bolt material and even the bottom and side plate and safe due to this load.

- From the structural analysis for the given loading condition, it is evident that the maximum stress on the PV-235 Mainframe is below the Yield stress on the material, so the design is safe under the given loading.
- Reaction Forces applied in the Ansys tool matches well with the given input.

REFERENCES

[1]. Dario Croccolo – Massimiliano De Agostinis – Nicolò Vincenzi: ' Structural Analysis of an Articulated Urban Bus Chassis via FEM: a Methodology Applied to a Case Study', Strojniški vestnik - Journal of Mechanical Engineering 57(2011)11, 799-809.

[2]. Ahmad O. Moaaz and Nouby M. Ghazaly: 'FINITE ELEMENT STRESS ANALYSIS OF TRUCK CHASSIS USING ANSYS: REVIEW', International Journal of Advances in Engineering & Technology, Nov., 2014. Vol. 7, Issue 5, pp. 1386-1391.

[3]. Sairam Kotari, V.Gopinath: ' STATIC AND DYNAMIC ANALYSIS ON TATRA CHASSIS', International Journal of Modern Engineering Research (IJMER), Vol.2, Issue.1, pp-086-094.

[4]. S.Prabakaran, K.Gunasekar: ' Structural Analysis of Chassis Frame and Modification for Weight Reduction', INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY, ISSN: 2277-9655, May, 2014.

[5]. Swami K.I., Prof. Tuljapure S.B.: 'Analysis of Ladder Chassis of Eicher 20.16 Using FEM', IOSR Journal of Applied Geology and Geophysics (IOSR-JAGG), e-ISSN: 2321–0990, p-ISSN: 2321–0982.Volume 2, Issue 1 Ver. I. (Jan. 2014), PP 06-13.

[6]. Madhu Ps and Venugopal T R: 'Static Analysis, Design Modification and Modal Analysis of Structural Chassis Frame', Madhu Ps et al Int. Journal of Engineering Research and Applications, ISSN: 2248-9622, Vol. 4, Issue 5(Version 3), May 2014, pp.06-10.

[7]. Frederico Mol Alvares da Silva, Luiz Felipe Adão Torre e Silva, Marcelo de Bittencourt Couto e Mello: ' MODAL TESTING OF A TUBULAR VEHICLE CHASSIS', 20th International Congress of Mechanical Engineering, November 15-20, 2009, Gramado, RS

[8]. Mahvi Malik Shahzad ,Shaikh Rizwan,Ansari Mudassir: 'Finite Element Modal Analysis Of

[9]. 4x2 Truck Chassis Using Pro/Mechanica', International Journal of Engineering, Education and Technology (ARDIJEET) ISSN 2320-883X, VOLUME 3, ISSUE 2, 01/04/2015.

[10]. Dr.R.Rajappan, M.Vivekanandhan: 'Static and Modal Analysis of Chassis by Using Fea', The International Journal Of Engineering And Science (Ijes), Volume2, Issue 2, Pages 63-73, 2013.

[11]. A.Hari Kumar, V.Deepanjali: 'Design & Analysis Of Automobile Chassis', International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 5, Issue 1, January 2016.

[12]. Amit Patil, Amol Kolhe: 'Review on Engineering Failure Analysis of Various Mechanical Systems', International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056, Volume: 02 Issue: 02 | May-2015, p-ISSN: 2395-0072.

[13]. Vivekanand Agalagatti, Dr. M A Kamoji, S Muthukumar: 'Study on Theoretical and Numerical analysis of Main frame- A Structural part in Backhoe Loader', International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 02 Issue: 05 | Aug-2015, p-ISSN: 2395-0072.

Finite Element Structural Analysis of a Pit Viper 235 Main Frame © 2021 by Dr. Anand Sn, Dr. Manohar Hs, Kumar Thanaiah is licensed under CC BY-NC-ND 4.0