Static Structural Analysis of the Sports Utility Vehicles Patriot Chassis

W Aryadi\(^1\), Kriswanto\(^1\), A Roziqin\(^1\), and J Jamari\(^2\)

\(^1\)Department of Mechanical Engineering, Universitas Negeri Semarang, Gd E9 Kampus Sekaran Gunungpati, Semarang, Indonesia
\(^2\)Department of Mechanical Engineering, University of Diponegoro, Jl. Prof. Sudharto Kampus UNDIP Tembalang, Semarang, Indonesia

widyaaryadi@mail.unnes.ac.id

Abstract. This paper presents a static structural analysis of the Sports Utility Vehicles (SUV) Patriot chassis uses a type of ladder frame with 440 G JIS STAM carbon steel tube and bracket material is AISI 1020. Static load on the patriot chassis is varied in a normal load and an extreme load. The element method simulation is carried out on the chassis until the horizontal chassis position is 0 \(^\circ\), 30 \(^\circ\) downhill and 30 \(^\circ\) uphill. Static structural analysis on the patriot chassis frame using the finite element method found that the overall design is safe under various load variations and support variations. Applied normal load with a total of 4 supports (4 wheels) on the patriot chassis design which is varied by its tilt, show that the 0 \(^\circ\) chassis position produces the highest von mises stress (86.1 MPa) and the highest displacement (0.38mm). The results of extreme loading on various supports of the 3 wheels alternately patriot chassis design show that the highest von mises stress (264 MPa) and the highest displacement (5.11 mm) occurs at 0 \(^\circ\) chassis position with rear right not support. The factor of safety from various variations in loaded, tilt position, and the number of supports of the Patriot chassis design is more than 1 so that the chassis design is declared safe.

1. Introduction

Patriot is the name of a prototype 1000cc petrol-engine car developed by Universitas Negeri Semarang. This prototype is made using components produced by the automotive component medium industry in Indonesia. One of the components designed by the UNNES research team is a chassis frame that made by PT. Karya Paduyasa Tegal. Patriot has been developed in two types, sports utility vehicles and pickup trucks. Both types of Patriots car require a strong and safe chassis to support vehicle, passenger and cargo loads. Chassis components need to be developed because previously it was only used for the pickup type. The position of the bracket, the shape of the bracket and the load data on the chassis design of the SUV model are different from the type of pickup, so it is necessary to analyze the structural strength to ensure the design is safe to use.

The chassis is the most important element that provides strength and stability to a vehicle in various conditions [1]. The chassis serves as a framework for supporting the body and different parts of the vehicle. The chassis type that is used in Patriot Prototype car is ladder frame. A ladder frame is the oldest form of automotive chassis still used in most SUVs [2]. There are many studies that have been conducted recently dealing with stress analysis on chassis of various vehicles using finite element
method (FEM) [2-15]. Stress analysis of the low loader chassis obtained that the location of maximum deflection and maximum stress in accordance with the theoretical maximum location of a simple beam under a uniformly distributed load [5]. To meet the design and use requirements can be seen that there is no excessive stress concentration, and the carrying capacity is greatly increased [6]. Stress analysis of dump truck chassis using finite element method shows that the analysis can be used to predict the bending and torsion stresses of frames [13]. Finite element analysis in the design of the go-kart chassis is used to analyze deformation and equivalent stress occurring on the structure on the application of forces on it. Finite element analysis in the design of the go-kart chassis is used to analyze deformation and equivalent stress occurring on the structure on the application of forces on it [14]. Stress analysis is performed in order to ensure that the design will meet its intended function in a given loads environment.

There are many studies on static structure analysis on chassis design [2-15], there is no chassis design for this type of SUV which is given extreme loaded. This work presents stress analysis and displacement of the chassis design under normal and extreme load uses a using finite element method. Patriot prototype car SUV type is a vehicle that is planned to be able to cross tough terrain in rural and mining areas, so extreme loading is one that needs to be investigated.

2. Method

Static structural analysis on the patriot chassis frame using the finite element method with the help of CATIA V5. Finite Element method is numerical method to finding an accurate solution of complex mathematical and structural problems, this method is the actual structure replaced by parts or elements [16]. The chassis design of the patriot SUV type is made in a 3D model using CATIA V5 as shown in Figure 1. Applied material properties to the 3d model according to table 1 where the pipe and tube materials use JIS G 3472 STAM 440 G carbon steel while the support bracket uses AISI 1020 Carbon Steel.

| Material properties | Pipe and Tube | Brackets |
|---------------------|--------------|----------|
| Yield Strength (MPa) | 305          | 305      |
| Tensile Strength (MPa) | 440         | 420      |
| Density [kg/m³]   | 7.8          | 7.87     |
| Elongation%       | 25           | 15       |
| Young Modulus (GPa) | 215          | 205      |

Figure 1. The 3D model of the Patriot chassis frame
Static loads on the Patriot SUV type chassis consist of engine load, transmission load, body load, other component loads, passenger loads and payload loads. Load data is presented in Table 2. There are two types of load applied, namely normal load and extreme load, which is three times the normal load. The loading is given in normal and extreme conditions, where the extreme load is assumed when the vehicle experiences a heavy load when the road is potholes and passing steep rocky roads. The chassis position is varied in three kinds of terrain conditions; horizontal, uphill and downhill.

The application of mesh type on the patriot prototype chassis model is tetrahedron with absolute size 10mm, 31,167 of nodes number, and 91,532 elements number. The constraint is placed on the support, namely the suspension bracket with a variation of loading according to Table 3. A variation of the support is also carried out where one of the wheels is alternately removed and given an extreme load. The applied load to the patriot chassis is shown in Figure 2.

### Table 2. Loads applied to the Patriot chassis frame

| Loads              | Support                        | Normal (N) | Extreme (N) |
|--------------------|--------------------------------|------------|-------------|
| Engine, transmission| Bracket engine & transmission  | 1260       | 3,780       |
| Body, others part  | Bracket body                   | 2000       | 12,300      |
| Passengers (8)     | Bracket body                   | 7200       | 21,600      |
| Others Load (cargo)| Bracket body                   | 1900       | 5,700       |

### Figure 2. Applied load to the Patriot chassis

### Table 3. Static load variations

| Chassis position | Load Type | Constraints                                      |
|------------------|-----------|--------------------------------------------------|
| Normal 0°        | Normal    | 4 wheels (4 support)                             |
| Normal 0°        | Extreme   | 3 wheels (3 support) alternately                 |
| Downhill 30°     | Normal    | 4 wheels (4 support)                             |
| Downhill 30°     | Extreme   | 3 wheels (3 support) alternately                 |
| Uphill 30°       | Normal    | 4 wheels (4 support)                             |
| Uphill 30°       | Extreme   | 3 wheels (3 support) alternately                 |

If the allowable strength is greater than the applied stress, then the part will not fail [18]. The design will be accepted if the factor of safety, $FoS$, is more than 1. If it is less than 1, the design is considered failed. Factor of safety calculation uses equation (1), where $FoS$ is factor of safety, $S_{allow}$ is allowable strength in MPa, and $\sigma_{app}$ is applied stress in MPa.

$$FoS = \frac{S_{allow}}{\sigma_{app}}$$ (1)

### 3. Results and Discussion

Patriot chassis frame design has been simulated on various loading variations using static structural analysis in two types of conditions, namely normal loads and extreme loads.
### 3.1. Static structural analysis under normal load

Normal loading is applied to the chassis with four supports (4 wheels) in the straight 0°, 30° downhill, and 30° uphill chassis positions. Von mises stress, displacement and safety factor analysis results are presented in Table 4. The highest stress von of the three variations of the chassis tilt position is 0° chassis position. Meanwhile, the lowest von mises stress value is generated from the chassis simulation at 30° downhill chassis positions.

Figure 3 shows von mises stress at 0° chassis tilt. The simulation results show that the red area is the critical area with the highest von mises stress value (86.1 MPa), this value is below the yield strength value (305 MPa). The safety factor at the 0° chassis position is 3.5 which is calculated using equation 1. The safety factor from the calculation of the three designs above is 1, so the chassis design in the straight, downhill and uphill positions is declared safe.

| No. | Position (°) | Constraint | \( \sigma_{\text{von}} \) (MPa) | FoS | \( \delta \) (mm) |
|-----|--------------|------------|---------------------------------|-----|-----------------|
| 1.  | 0            | All support (4 wheels) | 86.1 | 3.5  | 0.38 |
| 2.  | 30           | All support, downhill   | 31.56 | 9.7  | 0.35 |
| 3.  | 30           | All support, Uphill     | 41.47 | 7.4  | 0.33 |

**Figure 3.** Von mises stress at 0° chassis position

The highest von mises stress in the 0° chassis position is on body bracket of the rear chassis end. The displacement simulation results are shown in Figure 4. The highest displacement value is at the chassis position 0° (0.384 mm) which is shown in the red vector, which is at the rear end of the chassis frame. This section has the largest displacement because this section is subjected to large loads and is far from the support.

### 3.2. Static structural analysis under extreme load

Extreme loading is given to a chassis with a 3 wheels support as in Table 3, where the load data follows Table 2. The support is varied with one of the wheels removed alternately in the straight chassis position 0°, 30° downhill and 30° uphill. Von mises stress value and displacement simulation results are presented in Table 5 and the safety factor is calculated using equation (1). The highest Von mises stress from various variations in chassis tilt and support positions is the 0° chassis position with rear right not support.
Whereas the lowest von mises stress value is generated from the chassis simulation at 30° downhill position with rear left not support. Figure 5 shows von mises stress at 0° chassis tilt with rear right not support. The simulation results show that the red area is the critical area with the highest von mises stress value (264 MPa), this value is below the yield strength value (305 MPa). The safety factor at the 0° chassis position is 1.16 which is calculated using equation 1. The safety factor from the calculation of all variations gets a value above 1, so the chassis design in the horizontal, downhill, and uphill positions on various supports with extreme loads is declared safe.

![Figure 4. Displacement at 0° chassis position](image)

**Figure 4.** Displacement at 0° chassis position

**Table 5.** Results of the static structural analysis

| No. | Tilt position (°) | Constraint                      | $\sigma_{von}$ (MPa) | $FoS$ | $\delta$ (mm) |
|-----|------------------|---------------------------------|----------------------|-------|---------------|
| 1.  | 0                | Rear Left Not Support           | 258                  | 1.18  | 4.5           |
| 2.  | 0                | Rear Right Not Support          | 264                  | 1.16  | 5.11          |
| 3.  | 0                | Front Left Not Support          | 258                  | 1.18  | 2.2           |
| 4.  | 0                | Front Right Not Support         | 257                  | 1.19  | 2.1           |
| 5.  | 30               | Rear Left Not Support, downhill | 94                   | 3.2   | 1.72          |
| 6.  | 30               | Rear Right Not Support, downhill| 113                  | 2.7   | 1.87          |
| 7.  | 30               | Front Left Not Support, downhill| 128                  | 2.4   | 4.96          |
| 8.  | 30               | Front Right Not Support, downhill| 219                  | 1.4   | 4.59          |
| 9.  | 30               | Rear Left Not Support, Uphill   | 124                  | 2.5   | 1.81          |
| 10. | 30               | Rear Right Not Support, Uphill  | 124                  | 2.5   | 1.9           |
| 11. | 30               | Front Left Not Support, Uphill  | 198                  | 1.5   | 4.53          |
| 12. | 30               | Front Right Not Support, Uphill | 126                  | 2.4   | 4.14          |
4. Conclusion

Static structural analysis on the patriot chassis frame using the finite element method software found that the overall design is safe under various load variations and support variations. Applied normal load with a total of 4 supports (4 wheels) on the patriot chassis design which is varied by its tilt, so that the 0° chassis position produces the highest von mises stress (86.1 MPa) and the highest displacement (0.38mm). The highest stress is found on the body bracket from the rear end of the chassis, as well as in the highest displacement at the rear end of the chassis. The displacement is the farthest position from the support.

The highest displacement value is at 0° chassis position (5.11mm) which is displayed in the red vector, which is on the right side of the rear end of the chassis frame. This section has the largest displacement because this section is subjected to large loads and is far from the support.

Figure 6. Displacement at 0° chassis position with rear right not support

Figure 5. Von mises stress at 0° rear right not support

Figure 5 shows that the highest von Mises stress is in the 0° chassis position with rear right not support, as shown in the red area, namely on the left suspension support bracket. This section is a support that receives a high load because the support on the right is released, while the displacement simulation results are shown in Figure 6.

The highest displacement value is at 0° chassis position (5.11mm) which is displayed in the red vector, which is on the right side of the rear end of the chassis frame. This section has the largest displacement because this section is subjected to large loads and is far from the support.

4. Conclusions

Static structural analysis on the patriot chassis frame using the finite element method software found that the overall design is safe under various load variations and support variations. Applied normal load with a total of 4 supports (4 wheels) on the patriot chassis design which is varied by its tilt, so that the 0° chassis position produces the highest von mises stress (86.1 MPa) and the highest displacement (0.38mm). The highest stress is found on the body bracket from the rear end of the chassis, as well as in the highest displacement at the rear end of the chassis. The displacement is the farthest position from the support. The results of extreme loading on various supports of the 3 wheels alternately patriot chassis design shows that the highest von mises stress (264 MPa) and the highest displacement (5.11 mm) occurs at 0° chassis position with rear right not support. The highest stress area is on the left suspension support bracket and the highest displacement is on the right side of the rear end of the chassis frame. The displacement in this position is greatest because it has only one support at the rear and the load is given 3 times the normal load (extreme load).
The factor of safety from various variations in loaded, tilt position, and the number of supports shows the factor of safety is more than 1, so that the chassis design is declared safe.

References

[1] Vijayan S N, S Karthik, K Maharaja 2016 Finite Element Analysis of Automotive Chassis with Various Cross-Section, *International Conference on Systems, Science, Control, Communication, Engineering and Technology* 2 201-205

[2] Jena J, Chakraborty A, and Jain V 2018 Structural Analysis of Ladder Type Suv Chassis Frame (Diagonal Cross Bracing) using ANSYS 14.0 *GRDJE* 3 28-45

[3] Kepka M and Miloslav KJ 2020 Accelerated fatigue testing on special tracks as new part of methodology for bus/trolleybus development *Engineering Failure Analysis* 118

[4] Avikal S, Bisht A, Sharma D, Hindwan H, Yadav S, Kumar KCN, Thakur P 2020 Design and fatigue analysis of front axle beam of a heavy duty truck using ansys *Materials Today: Proceedings* 26 3211-3215

[5] Nor M A M, Rashid H, Mahyuddin W M F W, Azlan M A M, and Mahmud J 2012 Stress Analysis of a Low Loader Chassis *Procedia Engineering* 41 995-1001

[6] Yanhong C and Feng Z 2011 The Finite Element Analysis and The Optimization Design of The Yj3128-type Dump Truck’s Sub-Frames Based on ANSYS *Procedia Earth and Planetary Science* 2 133 – 138

[7] Ingole N K, and Bhope D V 2011 Stress analysis of tractor trailer chassis for self weight reduction *IJEST* 3 7218-7225

[8] Bhowmick P, Malhotra D, Agarwal P, and Ravi K 2017 Design and Analysis of a Pick-up Truck Chassis *IJISET* 1

[9] Ren Y, Yu Y, Zhao B, Fan C, and Li H 2017 Finite Element Analysis and Optimal Design for the Frame of SX360 Dump Trucks *Procedia Engineering* 174 638 – 647

[10] Chaundhary AR, Tushar S, and Raghendra KM 2015 Design and Development of Foldable Kart Chassis configuration *Materials Today: Proceedings* 2 1707-1713

[11] Arby J, Christophe M, Sebastien W, and Didier T 2018 Enhanced Fatigue Structural Stress Analysis Of A Heavy Vehicle Seam Welded Steel Chassis Frame: FEA Model Preparation, Weld Model Description, Fatigue Stress Calculation And Correlation With 10 Year Operating Experience *Procedia Engineering* 213 539-548

[12] Armentani E, Venanzio G, Antonio P, and Mauro P 2020 Design for NVH: topology optimization of an engine bracketsupport *Procedia Structural Integrity* 26 211-218

[13] Asker H K, Dawoodl TS, and Said A F 2012 Stress analysis of standard truck chassis during Ramping on block using finite element method: ARPN *Journal of Engineering and Applied Sciences* 7 641-648

[14] Prakash JS, Gankidi GR and Kavvampelly ST 2018 Numerical investigation on vibration characteristics and structural behaviour of different go-kart chassis configuration *Materials Today: Proceedings* 213 539-548

[15] Chandru BT and Suresh PM 2017 Finite Element and Experimental Modal Analysis of Car Roof with and without Damper *Materials Today Proceedings* 4 11237-11244

[16] JIS G 3472 STAM 440 G material properties. ISTW standart product, [http://www.istw.co.id/product/mechanical_tube/jis_g_3472_stam.html](http://www.istw.co.id/product/mechanical_tube/jis_g_3472_stam.html).

[17] AISI 1020 Carbon Steel Material properties, [https://www.efunda.com/materials/alloys/carbon_steels/show_carbon.cfm?id=aisi_1020&prop=all&page_title=aisi%201020](https://www.efunda.com/materials/alloys/carbon_steels/show_carbon.cfm?id=aisi_1020&prop=all&page_title=aisi%201020).

[18] Ullman D G 1984 *The Mechanical Design Process* (New York: McGraw Hill)