JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

Structural Strength Evaluation of Cable Type Automotive Window Regulator

[1]Prajwalkumar V Kunnal, [2]Dr Kirthan L J, [3]Sangamesh Akki
[1]PG Student, [2]Associate Professor, [3]PG Student
[1]Department of Mechanical Engineering,
[1]RV College of Engineering, Bangalore, Karnataka, India

Abstract : The automotive industry continually strives to enhance safety, performance, and reliability, with a focus on every component, including the window regulator. This study delves into the finite element modeling and structural assessment of window regulators, a critical yet often overlooked component. Through comprehensive analysis and simulation, the structural integrity of window regulators is evaluated under various conditions, including temperature extremes and load scenarios. The research employs state-of-the-art materials, advanced design techniques, and rigorous testing to ensure safety and functionality. This investigation not only provides valuable insights into the performance of window regulators but also contributes to the ongoing improvement of automotive components, aligning with industry demands for higher standards of quality and reliability.

Keywords: Structural analysis, Stress distribution, Deformation, Mesh quality Checks

1. INTRODUCTION

A window regulator is an integral component in automotive design responsible for controlling the movement of windows in vehicles. Its primary function is to enable passengers to raise or lower the windows with ease, providing essential ventilation, temperature control, and convenience within the vehicle's cabin. A single lift cable-type window regulator is a mechanical component designed for use in automotive applications. Its primary function is to control the movement of a vehicle's window glass, allowing it to be raised or lowered smoothly with the assistance of a cable mechanism. This type of window regulator typically consists of a series of pulleys, cables, and a motor that work in tandem to move the window glass. It is a crucial part of a vehicle's power window system, providing convenience and control to the driver and passengers. The structural strength evaluation of a car window regulator is a crucial step in ensuring the safety and reliability of the power window system. The evaluation process involves testing the regulator's structural integrity and durability under different operating conditions, such as temperature, different load conditions and abusive working conditions. Static structural analysis of a window regulator is a crucial step in the design and development process for several reasons. First and foremost, it ensures the safety of vehicle occupants. The window regulator plays a vital role in maintaining the structural integrity of the vehicle's window. By subjecting the regulator to static load scenarios, engineers can determine whether it can withstand forces such as wind pressure, external impacts, or the weight of the window itself. Ensuring that the window regulator remains stable and secure under these conditions is essential for preventing accidents, injuries, or the potential ejection of passengers during a collision. The structural strength of a window regulator is a critical factor that determines its ability to withstand external forces and stresses. A window regulator is responsible for controlling the movement of a vehicle's windows and must be able to handle various loads and forces that act on it during operation. The evaluation helps to identify critical areas of the window regulator that are prone to failure and propose design modifications to enhance its performance. By improving the structural strength of the window regulator, its reliability and durability are increased, leading to improved safety and performance of vehicles.

2. OBJECTIVES OF THE WORK

The objectives of the current work are.

1. Evaluate the structural performance of the window regulator to ensure it can withstand various loads and environmental conditions without failure.
2. Determine the stress and strain distribution within the components of the window regulator to identify potential weak points and areas of concern

3. METHODOLOGY

Steps followed to perform the project work is as follows.

1. Literature survey on window regulators
2. CAD design of Window regulator model in CATIA V5
3. Material Selection
4. FE Model preparation in HYPERMESH
5. Perform structural analysis using ABAQUS.
6. Analysis of the result

7. Identify areas of improvement.

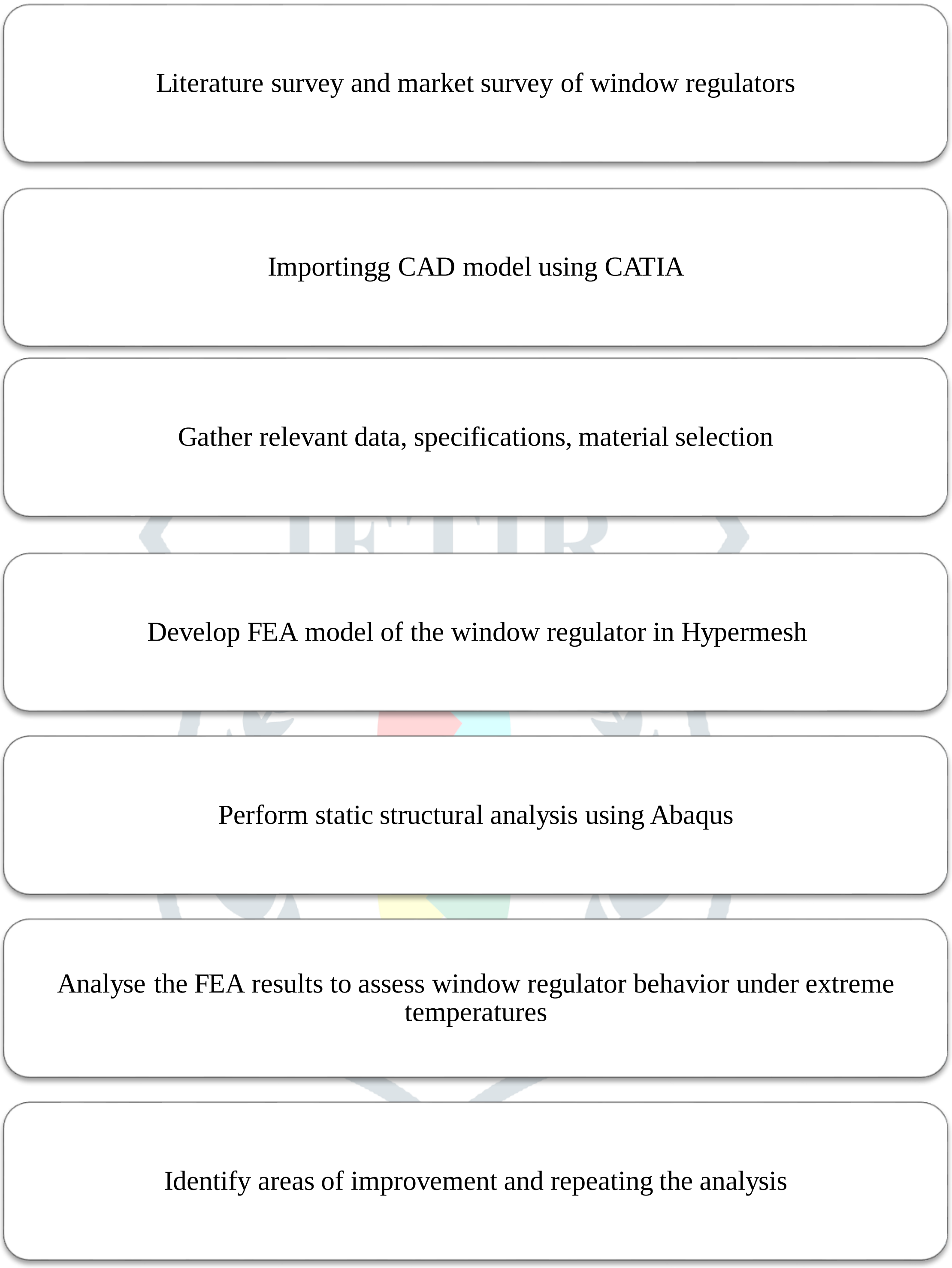


Fig 3.1 Methodology steps to conduct the project

3.1 LITERATURE SURVEY

Analysis type	Details of the Analysis
Dynamic analysis of window regulator using LS DYNA	In this analysis, the focus was on a specific type of X arm window regulator design, which employs a pin joint mechanism. • To accurately capture the window glass's ascent speed, it was needed to analyse the phenomenon over a duration of 3 to 4 seconds. • However, when utilizing dynamic finite element analysis methods like LS-DYNA, a significant number of iterations can lead to a substantial increase in CPU time. To mitigate this, it was aimed to reduce the number of iterations, which in turn required increasing the time step size. Consequently, they adjusted the meshing to achieve a time step size of 1.05e-6 seconds.
Fatigue life analysis of window regulator using ABAQUS and FE-SAFE	In this analysis, both ABAQUS and FE-SAFE software packages were employed to conduct a comprehensive simulation analysis of the fatigue life of a locked-rotor glass regulator at its top dead centre position. • The fatigue life diagram derived from the analysis reveals that certain components, such as the slider shrapnel bending and the guide rail flanging, as well as the guide pulley around the mounting hole, exhibit relatively low fatigue life. • This suggests a potential risk of fatigue failure in the glass regulator. This approach establishes a theoretical foundation for the design considerations related to the fatigue life of the glass regulator.
Wear analysis of window regulator using ABAQUS	For the analysis, the 3D model of the window regulator using CATIA V5 software and then imported it into the FE software ABAQUS. • To account for the impact of the slider's sliding position along the guide rail on contact pressure, six critical locations were selected for analysis. These six key contact points on the slider experience higher contact pressures and are prone to wear due to initial interference. • An investigation was conducted into the contact pressure at these six key contact points (A1, A2, B1, B2, C1, C2) under different operational positions and when the bolt lifting moment is applied. Through the finite element analysis it was observed that the average contact pressure at points A1 and C1 exceeds particularly in positions where the initial interference between the slider and guide rail is most significant.
Static analysis of window regulator using ANSYS	This analysis presents an introduction to four crucial components and three critical operational scenarios of the glass regulator. • A single track rope-wheel glass regulator utilized. The essential components of this single track rope-wheel glass regulator encompass the guide rail, slider assembly, guide pulley, motor assembly, and upper and lower wire rope assembly. • The static analysis results of the glass regulator indicate the presence of stress concentration and significant strain primarily in the guide rail. Stress concentration was also observed in the slider, guide pulley, and motor plate

Table 4.1 Different types of analysis performed on window regulator

3.2 CAD MODELLING OF WINDOW REGULATOR

Computer-Aided Design (CAD) modeling is a crucial step in the structural analysis of a window regulator. Here's an explanation of the CAD modeling process for this purpose.

- Component Identification: First, identify and list all the individual components that make up the window regulator. These components may include the rail, panel plate, brackets, gears, and other relevant parts.

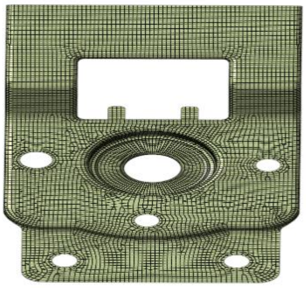
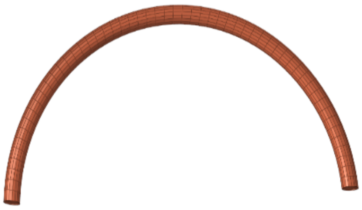
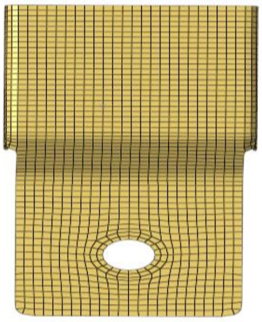
- CAD Software Selection: Choose a suitable CAD software package for the modeling process. Popular choices include SolidWorks, AutoCAD, CATIA, or Creo. Select software that best fits your organization's expertise and project requirements.
- 2D Sketching: Begin by creating 2D sketches of each component. These sketches serve as the foundation for the 3D models. Use tools within the CAD software to draw accurate representations of each part, specifying dimensions and geometric features.
- Extrusion and Revolving: Extrude or revolve the 2D sketches to create 3D models of the components. This step adds depth and volume to the parts, turning them into solid objects.
- Assembly Creation: Use the CAD software's assembly tools to assemble the components into a complete window regulator. Ensure that the assembly replicates the real-world arrangement of parts, including their relative positions and interactions.
- Material Assignment: Assign appropriate materials to each component within the CAD model. This step is essential for accurately simulating how the materials will behave under different load conditions during structural analysis.
- Boundary Conditions: Define the boundary conditions within the CAD model. Specify how the window regulator is fixed or supported in the analysis. This includes constraints, such as fixing certain points or edges, and defining applied loads or forces.
- Mesh Generation: Generate a mesh over the CAD model. Meshing divides the model's surfaces into smaller elements, facilitating the application of finite element analysis (FEA). Adjust the mesh density to balance accuracy and computational efficiency.
- Load Application: Apply the loads and forces that mimic real-world conditions. Depending on your analysis goals, these loads may include various scenarios, such as wind resistance, impact forces, or window glass weight.
- Solver Setup: Prepare the CAD model for FEA by setting up the solver parameters within the CAD software or exporting the model to FEA-specific software like ANSYS or Abaqus.
- Analysis: Run the structural analysis using FEA techniques. The software will simulate how the window regulator components respond to the applied loads, providing stress, strain, deflection, and deformation data.
- Results Visualization: Analyze and visualize the results of the structural analysis. Identify areas of high stress or strain, deformation patterns, and any potential failure points. Ensure that the design meets safety and performance criteria.
- Iterative Design: Based on the analysis results, make necessary design modifications to improve structural integrity or optimize performance. Iterate through the CAD modeling and analysis process until the desired outcomes are achieved.
- Documentation: Create detailed documentation of the CAD model, analysis setup, and results. This documentation is essential for validation, quality control, and future reference.
- CAD modeling for structural analysis is a vital step in ensuring the window regulator's reliability and safety. It allows engineers to assess the design's performance under various conditions and make informed decisions to enhance the product's structural integrity.

3.3 MATERIAL SELECTION

Material	Young's Modulus
Manganese Steel alloy	**265MPa**
Stainless Steel	**260MPa**
Poly oxy methylene	**60MPa**

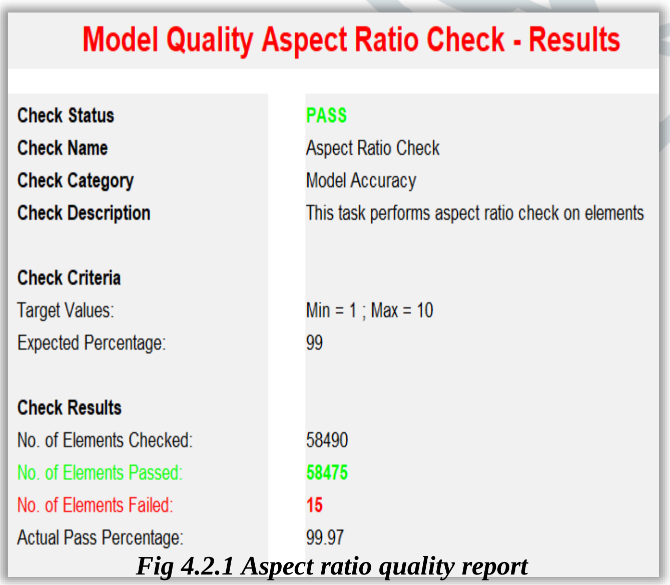
4.1 MESHED COMPONENTS

Component	Figure	Mesh details
Rail	 *Fig 4.11 Rail mesh*	Shell mesh 2D elements Quad4=S4R Tria3=S3R Element size: 1mm
Cursor	 *Fig 4.12 Cursor mesh*	Shell mesh 2D elements Quad4=S4R Tria3=S3R Element size: 1mm

Panel Plate	 *Fig 4.13 Panel plate mesh*	Shell mesh 2D elements Quad4=S4R Tria3=S3R Element size: 2mm
Cable	 *Fig 4.14 Cable mesh*	Hex mesh 3D elements Hex8=C3D8R Element size: 5mm
Bracket	 *Fig 4.15 Bracket mesh*	Shell mesh 2D elements Quad4=S4R Tria3=S3R Element size: 2mm

4.2 MESH QUALITY REPORTS

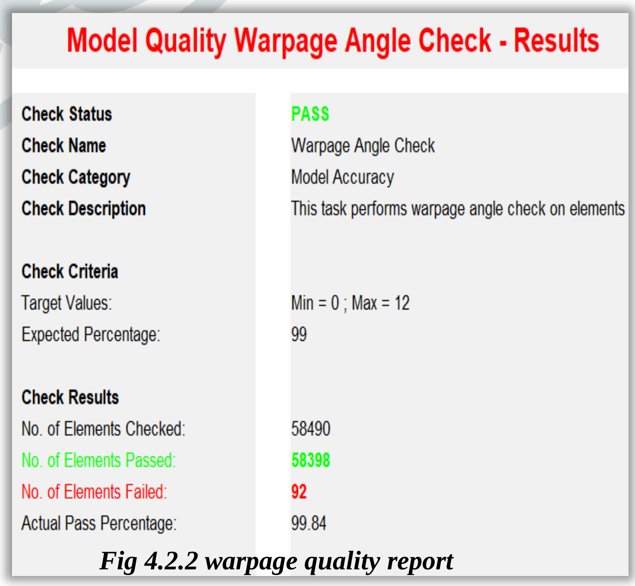
The different mesh quality reports for all the elements i.e 2D and 3D elements are described in this section.



Model Quality Aspect Ratio Check - Results

Check Status	PASS
Check Name	Aspect Ratio Check
Check Category	Model Accuracy
Check Description	This task performs aspect ratio check on elements
Check Criteria	
Target Values:	Min = 1 ; Max = 10
Expected Percentage:	99
Check Results	
No. of Elements Checked:	58490
No. of Elements Passed:	**58475**
No. of Elements Failed:	**15**
Actual Pass Percentage:	99.97

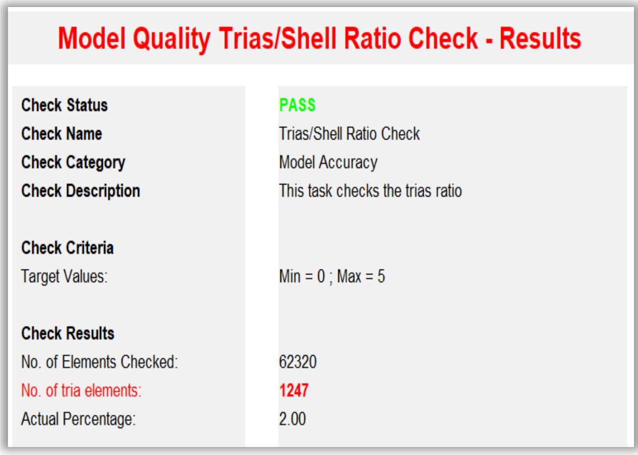
Fig 4.2.1 Aspect ratio quality report



Model Quality Warpage Angle Check - Results

Check Status	PASS
Check Name	Warpage Angle Check
Check Category	Model Accuracy
Check Description	This task performs warpage angle check on elements
Check Criteria	
Target Values:	Min = 0 ; Max = 12
Expected Percentage:	99
Check Results	
No. of Elements Checked:	58490
No. of Elements Passed:	**58398**
No. of Elements Failed:	**92**
Actual Pass Percentage:	99.84

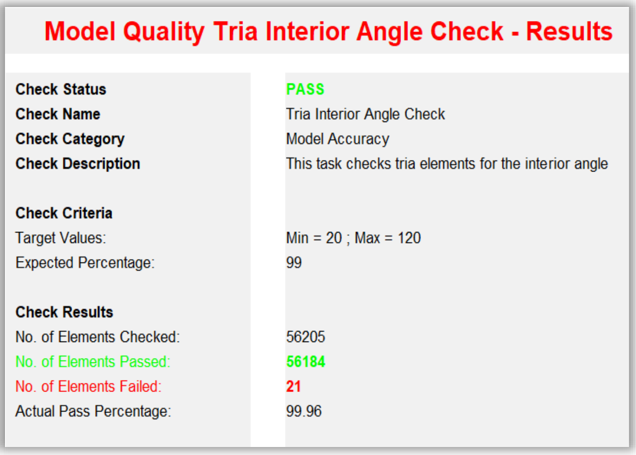
Fig 4.2.2 warpage quality report



Model Quality Trias/Shell Ratio Check - Results

Check Status	PASS
Check Name	Trias/Shell Ratio Check
Check Category	Model Accuracy
Check Description	This task checks the trias ratio
Check Criteria	
Target Values:	Min = 0 ; Max = 5
Check Results	
No. of Elements Checked:	62320
No. of tria elements:	**1247**
Actual Percentage:	2.00

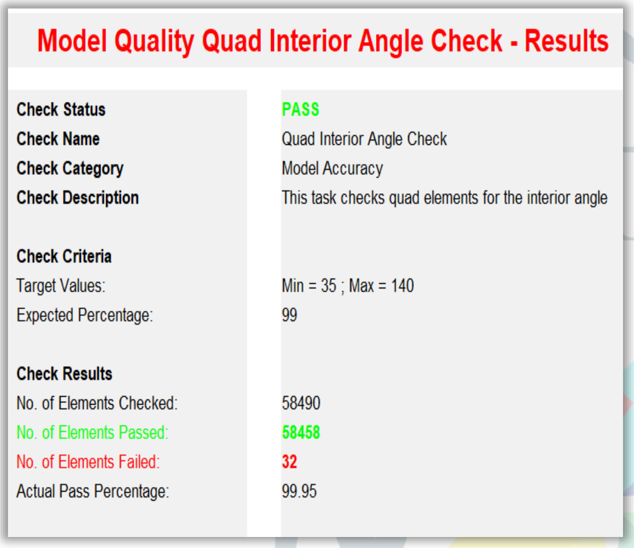
Fig 4.2.3 Tria/shell ratio quality report



Model Quality Tria Interior Angle Check - Results

Check Status	PASS
Check Name	Tria Interior Angle Check
Check Category	Model Accuracy
Check Description	This task checks tria elements for the interior angle
Check Criteria	
Target Values:	Min = 20 ; Max = 120
Expected Percentage:	99
Check Results	
No. of Elements Checked:	56205
No. of Elements Passed:	**56184**
No. of Elements Failed:	**21**
Actual Pass Percentage:	99.96

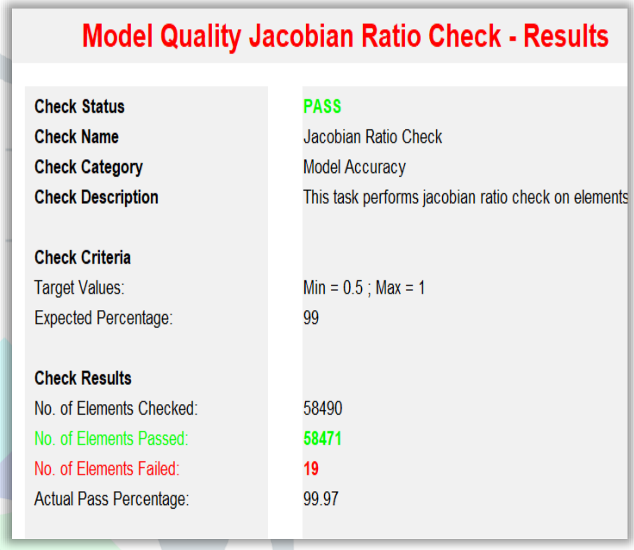
Fig 4.2.4 Tria interior angle quality report



Model Quality Quad Interior Angle Check - Results

Check Status	PASS
Check Name	Quad Interior Angle Check
Check Category	Model Accuracy
Check Description	This task checks quad elements for the interior angle
Check Criteria	
Target Values:	Min = 35 ; Max = 140
Expected Percentage:	99
Check Results	
No. of Elements Checked:	58490
No. of Elements Passed:	**58458**
No. of Elements Failed:	**32**
Actual Pass Percentage:	99.95

Fig 4.2.5 Quad interior angle quality report



Model Quality Jacobian Ratio Check - Results

Check Status	PASS
Check Name	Jacobian Ratio Check
Check Category	Model Accuracy
Check Description	This task performs jacobian ratio check on elements
Check Criteria	
Target Values:	Min = 0.5 ; Max = 1
Expected Percentage:	99
Check Results	
No. of Elements Checked:	58490
No. of Elements Passed:	**58471**
No. of Elements Failed:	**19**
Actual Pass Percentage:	99.97

Fig 4.2.6 Jacobian ratio quality report

5. RESULTS AND DISCUSSION

Table 5.1: Results for different load conditions

Results	***UST 23°C***		***UST 80°C***		***LST 23°C***		***LST 80°C***		***FG -40°C***		
	Rail	***PP***	***Rail***	***PP***	***Rail***	***PP***	***Rail***	***PP***	***Rail***	***PP***	***Bracket***
Deflection **(in mm)**	*2.6*	*1.6*	*2.5*	*1.6*	*1.5*	*0.9*	*1.6*	*0.7*	*7*	*2.6*	*3.7*
Stress **(in MPa)**	*256*	*258*	*262*	*260*	***266***	*251.68*	***271***	*259*	*263*	*260*	***273***

Table 5.1 Deflection and stress values for different loading condition

The structural analysis results for the window regulator components at various load conditions and temperatures reveal important insights. At room temperature (23°C), both upper and lower stall load conditions (UST and LST) exhibit stress levels within an acceptable range for the rail and panel plate (PP) components, indicating structural integrity. However, when subjected to extreme cold (-40°C) during the frozen glass load (FG) condition, the rail's stress exceeds the yield limit, warranting further investigation or design modifications. Interestingly, despite the elevated temperature (80°C) under UST, the rail's stress remains within acceptable limits. The deflection analysis shows that, in general, the components experience acceptable deflections, except for the bracket under the FG condition, where deflection and stress levels are notably high, suggesting a potential need for material changes or design adjustments to ensure safety and functionality. These findings underscore the importance of considering both temperature variations and load conditions in the structural evaluation of the window regulator to optimize its performance and reliability

6 CONCLUSION

The results obtained from the structural analysis of the window regulator components under various loading conditions and temperature settings provide a comprehensive overview of their performance.

- Under the Upper Stall Test (UST) at both 23°C and 80°C, the rail and panel plate exhibit minimal deflection, with values ranging from 1.5mm to 2.6mm, well within industry-specified limits. Stress levels remain comfortably within the acceptable yield limits, ensuring the design's safety and reliability for operational use.
- In the Lower Stall Test (LST) at 23°C and 80°C, the rail experiences slightly higher stress values but still maintains acceptable deflection levels. Despite the rail's stress slightly exceeding the yield limit, the observed deflection of 1.6mm remains within industry standards. The panel plate exhibits stress levels well within the acceptable range, further confirming the design's safety.
- The most critical scenario, the Frozen Glass Load Test (FG) at -40°C, reveals that the bracket experiences significantly higher stress levels, surpassing the yield limit and leading to a deflection of 3.7mm, which slightly exceeds industry-specified limits. To address this, it is advisable to consider altering the bracket's material or enhancing its design, potentially by increasing the fillet radius at bending regions.
- Overall, the window regulator design showcases excellent structural integrity and safety under most test conditions, with minor recommendations for enhancement in the bracket component under extreme cold conditions. These findings underscore the importance of rigorous testing and analysis to ensure the reliability and performance of automotive components in diverse environmental and load scenarios.

REFERENCES

1. T. Zahl, "All About Window Regulators & Motors," CARiD, 20, November 2014.
2. Mingzhang Chen et al, “Statistical analysis of automotive rope wheel glass regulator”, Journal of Physics: Conference Series, Volume 1654, August 2020.
3. Sarode, Er Jayesh Sudhakar, “Material and Structure Optimization of Car Door Window Regulator”, A case study, Academia.edu, 2017
4. Marcus Bohlin, Gunjan Nagpal, “Development of an Anti-Pinch System for Passenger Vehicles”, Department of Industrial and Materials Science, Chalmers University of Technology, Gothenburg, Sweden 2020.
5. A. Das, R. Mishra and K. Kumar, “Design of a five-bar linkage for an automated power window”, Materials Today: Proceedings, Volume 49, Part 2, 2022, Pages 433-439, Elsevier
6. Sergey Petkun. October 2018. Dynamic behavior of power window regulator system. profile/Sergey-Petkun/publication/328305605,
7. Mingzhang Chen *et al* 2020 *IOP Conf. Ser.: Earth Environ. Sci*. **571** 012109.
8. Xu, Y., Li, Y., & Li, C. (2021). Electric Window Regulator Based on Intelligent Control. *Journal of Artificial Intelligence and Technology*, *1*(4), 198–206.