Report - Homework Assignment 2

Sean Zimmari - SN 6151337

September 22, 2024

1 Introduction

The following report has the purpose to explain the adopted numerical methodology for the modeling of the structure supporting the wheel of the Perseverance Rover. The report is divide in 3 sections:

1. 'Section 2' - The computational method will model the effective structure with a truss structures, hence it is good practice to point out which are the most important assumptions when trusses are used for the simulation`\item` command.
2. 'Section 3' - A verification of the obtained results has been made using the software ABAQUS in order to evaluate the effective accuracy of the model
3. 'Section 4' - The final part will analyze the effects on the solutions if a rope is substituted to a truss element in the structure

2 Most important assumption

When truss elements are adopted, one has to be aware of the effects this adoption will lead. As a matter of fact, the most important caution is that **TRUSSES ARE ONLY ABLE TO SUPPORT AXIAL LOADINGS**, which are compression or traction.

This property has relevant effects on the result, since only in-plane deformations are considered. Consequently, the displacements calculate with the model will be overestimated than the effective ones because basically there is less stiffness. However, this is a conservative choice because if the structure was able to support these displacements, then it would be able to support the effective ones.

3 Verification in ABAQUS

Verifications in ABAQUS has been made in order to evaluate the accuracy of the model. After having modelled the truss structures, the following results have been produced:

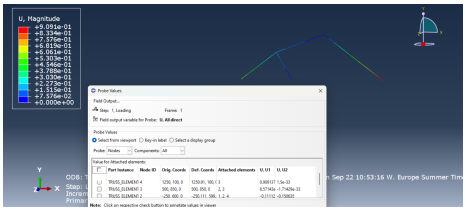

Figure 1: Displacements Verification in ABAQUS

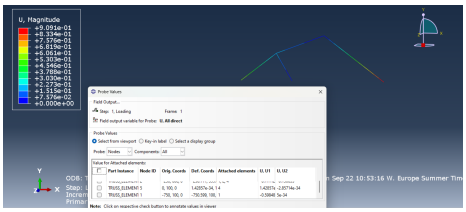

Figure 2: Displacements Verification in ABAQUS

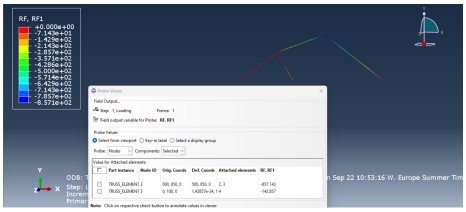

Figure 3: Reactions R1 Verification in ABAQUS

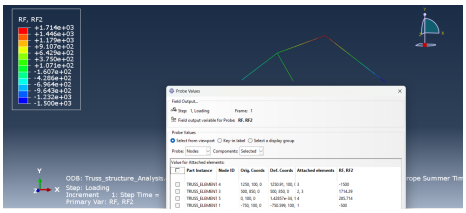

Figure 4: Reactions R2 Verification in ABAQUS

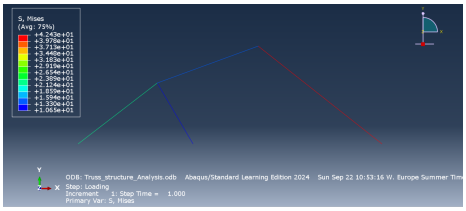

Figure 5: Stress Verification in ABAQUS

Looking at the results obtained with Python, one can conclude that the computational model is really accurate.

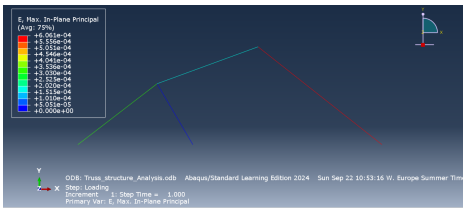

Figure 6: Strains Verification in ABAQUS

```
the stress in element  1  equals  -42.43 MPa
the stress in element  2  equals  10.65 MPa
the stress in element  3  equals  23.57 MPa
the stress in element  4  equals  -13.55 MPa
The displacement of node 1  in x-direction is  0.000 mm
The displacement of node 1  in y-direction is  0.000 mm
The reaction force at node 1  in x-direction is  -857 N
The reaction force at node 1  in y-direction is  1714 N
The displacement of node 2  in x-direction is  0.909 mm
The displacement of node 2  in y-direction is  0.000 mm
The reaction force at node 2  in x-direction is  0 N
The reaction force at node 2  in y-direction is  -1500 N
The displacement of node 3  in x-direction is  0.000 mm
The displacement of node 3  in y-direction is  0.000 mm
The reaction force at node 3  in x-direction is  -143 N
The reaction force at node 3  in y-direction is  286 N
The displacement of node 4  in x-direction is  -0.598 mm
The displacement of node 4  in y-direction is  0.000 mm
The reaction force at node 4  in x-direction is  0 N
The reaction force at node 4  in y-direction is  -500 N
The displacement of node 5  in x-direction is  -0.111 mm
The displacement of node 5  in y-direction is  -0.151 mm
The reaction force at node 5  in x-direction is  0 N
The reaction force at node 5  in y-direction is  0 N
```

Figure 7: Python Results

4 Hypothetical situation

Let's assume that a truss element is replaced with a rope in the structure. Unlike trusses, ropes **cannot support compression** thus its contribution to the global stiffness matrix will be null if the loading is compressive. In other words, its stiffness value will remain the same since it has same Young's modulus and cross sectional area, but in the case we have a compressive load to the element 2, then its contribution will be null in the global stiffness matrix. Hence, the code will have to check if the load applied to the element 2 is a compressive one, if so then it should avoid it in the global stiffness matrix.