UNIVERSITAT POLITÈCNICA DE CATALUNYA
BARCELONATECH
Escola d'Enginyeria de Barcelona Est

TREBALL FI DE GRAU

Grau en Enginyeria Mecànica

TENSION STUDY OF A CARBON FIBER BOGIE FRAME

Volum II

ANEXO

Autor: Abelardo J. Alfaro

Director: Antonio J. Sánchez Egea

Departament: DEM

Convocatòria: Gener 2021

Table of Content

List of Figures

List of Tables

16. List of Appendices

Welcome to the appendix.

Here you will find the additional documentation that will help with the interpretation and understanding of the bachelor's thesis of Mr. Alfaro. In this appendix, all of the content will be linked to the memory so that there is a connectivity between both of the documents.

a. Appendix I – Visual representation of each boundary conditions, followed by the result of each study case

Structural Steel S235 frame:

Case 1 - Vertical loads – Boundary conditions - Structure steel S235;

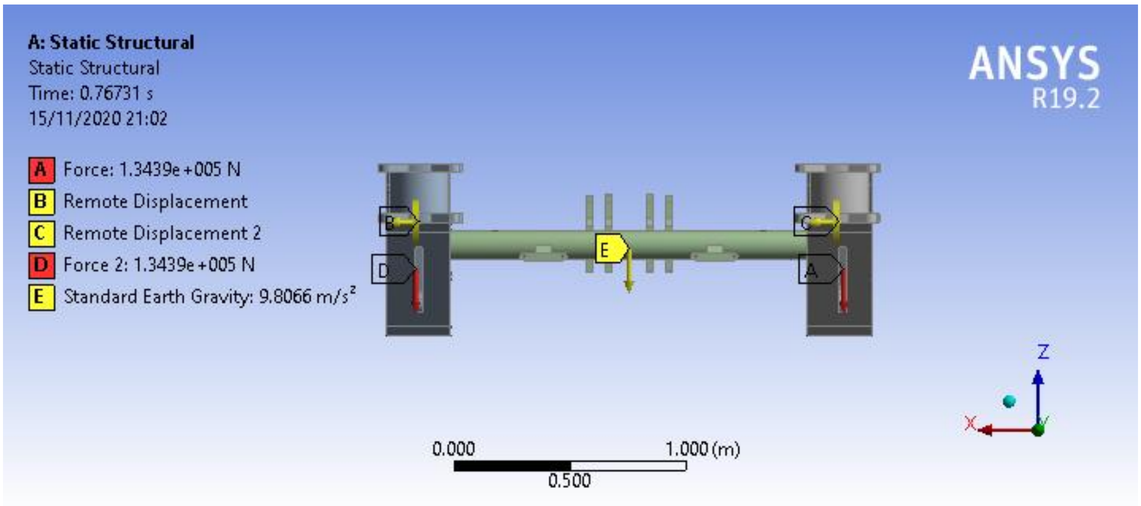


Figure 1. Boundary and forces for structure steel S235 case number 1 - view 1.

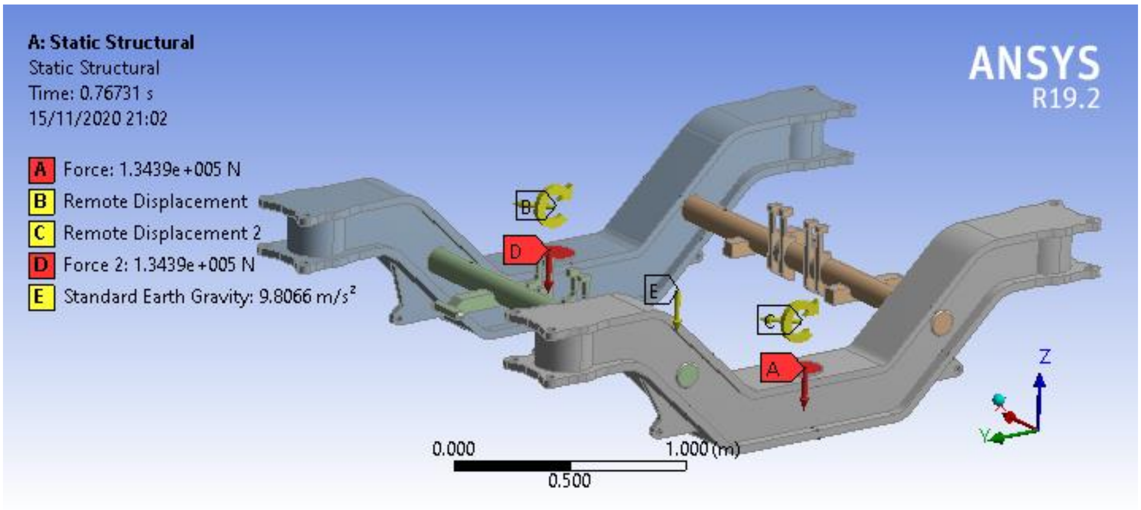


Figure 2. Boundary and forces for structure steel S235 case number 1 - view 2.

Table 1. Detailed table of loads and restrictions for case 1 of the Structure steel S235.

Definition				
Type	Force	Remote Displacement		Force
Define By	Vector			Vector
Magnitude	-1.3439e+005 N (ramped)			-1.3439e+005 N (ramped)
Direction	Defined			Defined
Suppressed	No			
X Component		0.01 m (ramped)		
Y Component		0.01 m (ramped)		
Z Component		0. m (ramped)		
Rotation X		0.1 ° (ramped)		
Rotation Y		0. ° (ramped)		
Rotation Z		0. ° (ramped)		
Behavior		Deformable	Coupled	
Rotation X			0.1 ° (ramped)	
Rotation Y			0. ° (ramped)	
Rotation Z			0. ° (ramped)	
Advanced				
Pinball Region		All		

Like calculated previously, the force would be a value of $F = 134387.2\text{ N} \cong \mathbf{134388}\ \boldsymbol{N}$ and the restrictions – (x, y, z) = (0.01, 0.01, 0) & (Rx, Ry, Rz) = (0.1, 0, 0).

Case 1 - Vertical loads – Results - Structure steel S235:

Total deformation:

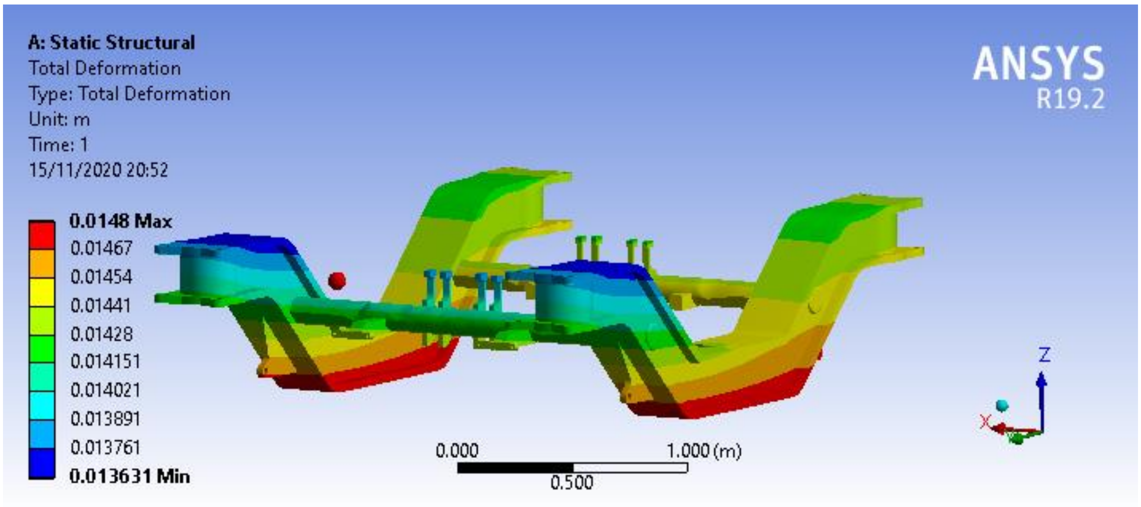


Figure 3. Total deformation 1 - structure steel - case 1

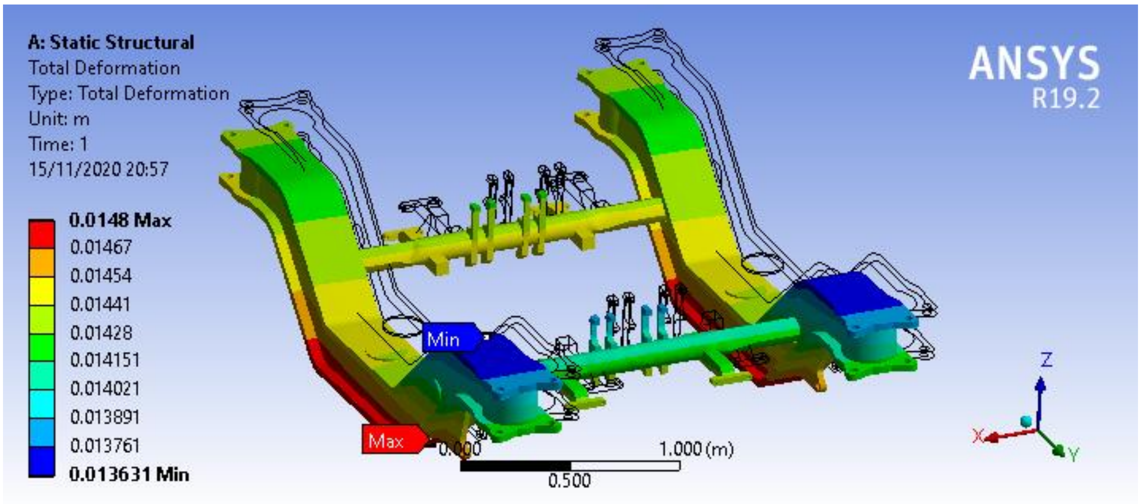


Figure 4. Total deformation 2 - structure steel - case 1

Von-Misses stress:

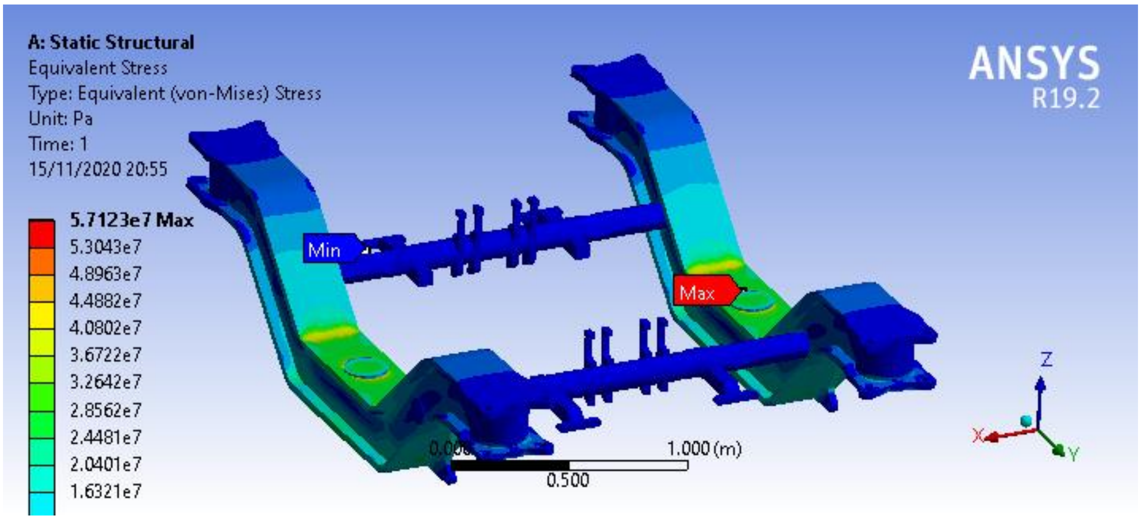


Figure 5. Von-Mises stress 1 - structure steel - case 1

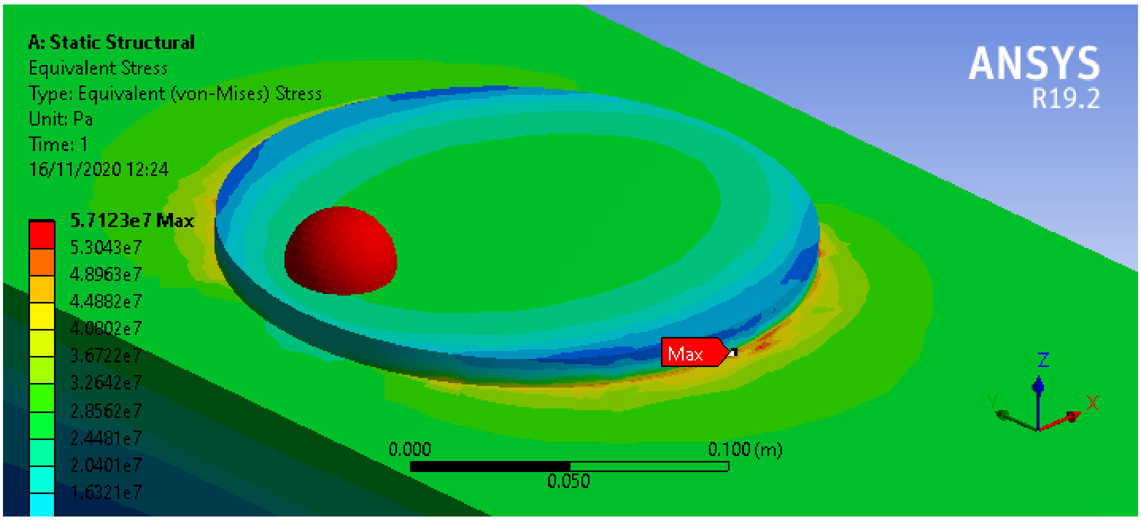


Figure 6. Von-Mises stress 2 - structure steel - case 1

Safety factor:

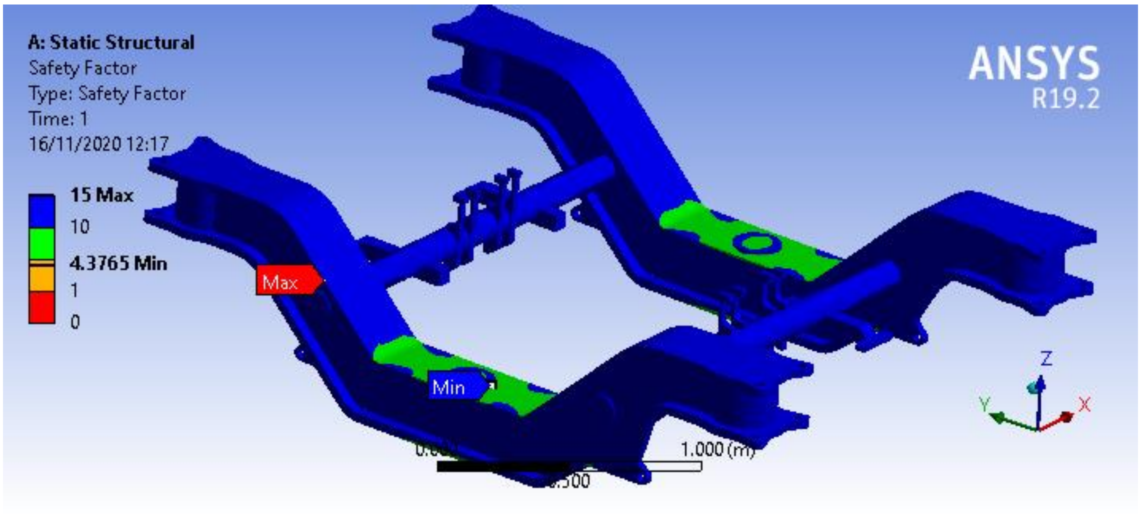


Figure 7. Safety factor 1 - structure steel - case 1

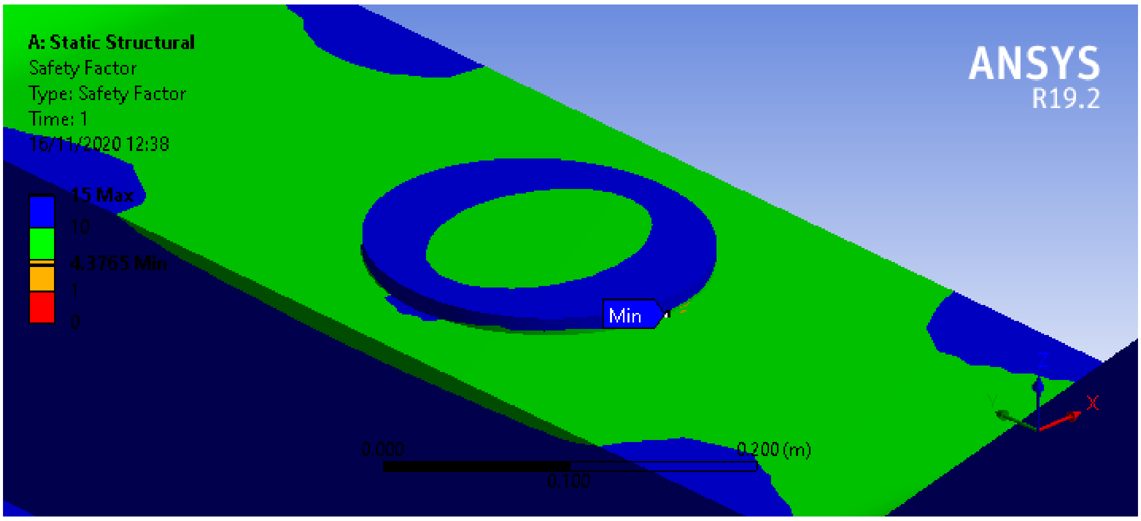


Figure 8. Safety factor 2 - structure steel - case 1

Case 2 - Lateral loads – Boundary conditions - Structure steel S235;

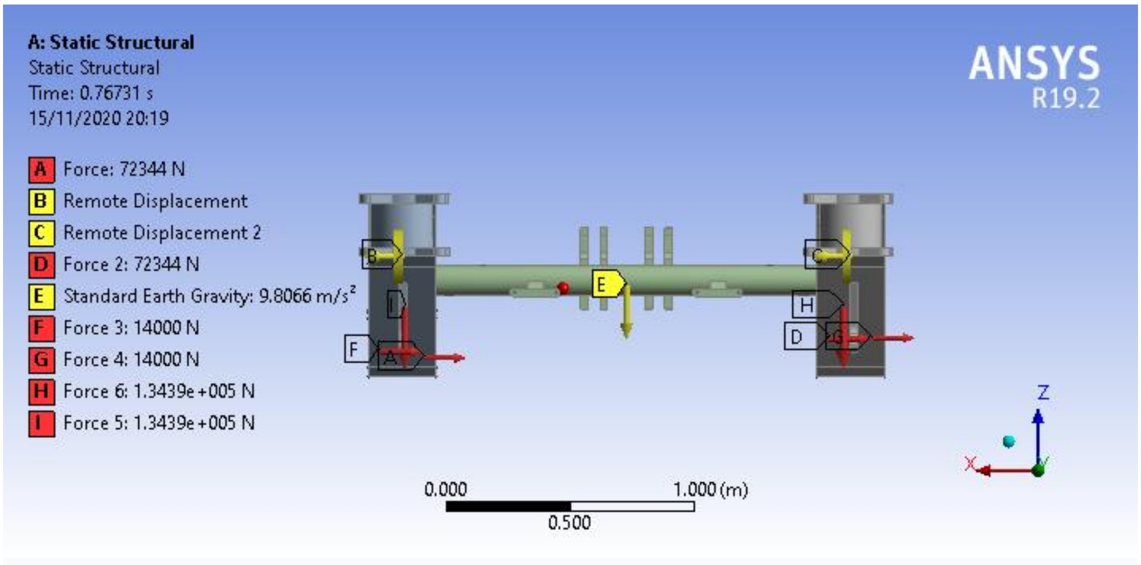


Figure 9. Boundary and lateral load for structure steel S235, view 1.

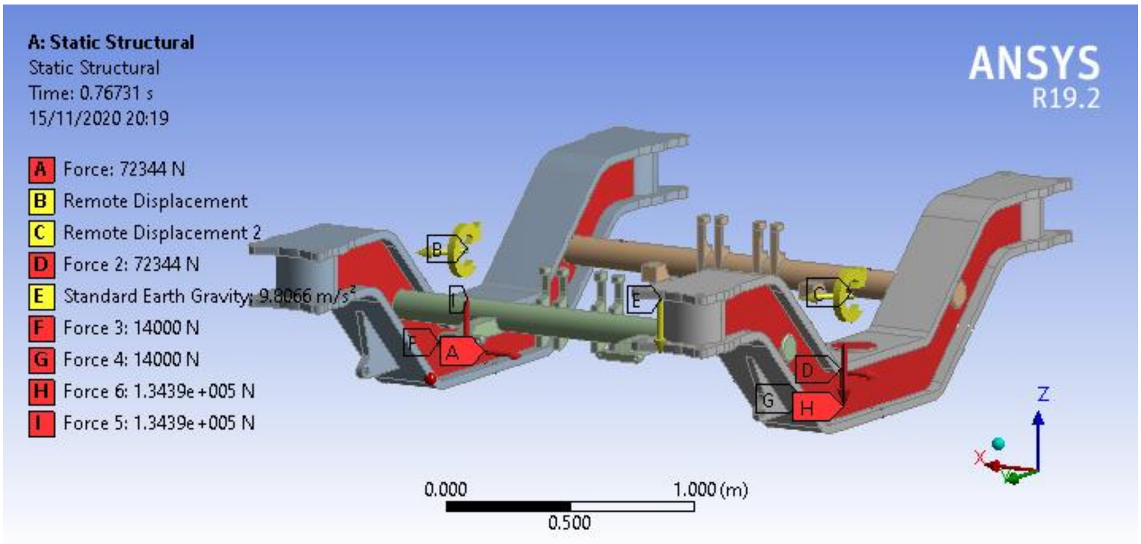


Figure 10. Boundary and lateral loads for structure steel S235, view 2.

Table 2. Detailed table of loads and restrictions for case 2 of the Structure steel S235.

<table>
<tr><th colspan="7">Definition</th></tr>
<tr><td>Type</td><td>Force</td><td colspan="2">Remote Displacement</td><td colspan="3">Force</td></tr>
<tr><td>Define By</td><td>Vector</td><td colspan="2"></td><td colspan="3">Vector</td></tr>
<tr><td>Magnitude</td><td>-72344 N (ramped)</td><td colspan="2"></td><td>-72344 N (ramped)</td><td>14000 N (ramped)</td><td>-1.3439e+005 N (ramped)</td></tr>
<tr><td>Direction</td><td>Defined</td><td colspan="2"></td><td colspan="3">Defined</td></tr>
<tr><td>Suppressed</td><td colspan="6">No</td></tr>
<tr><td>X Component</td><td></td><td colspan="2">0.01 m (ramped)</td><td colspan="3"></td></tr>
<tr><td>Y Component</td><td></td><td colspan="2">0.01 m (ramped)</td><td colspan="3"></td></tr>
<tr><td>Z Component</td><td></td><td colspan="2">0. m (ramped)</td><td colspan="3"></td></tr>
<tr><td>Rotation X</td><td></td><td>0.1 ° (ramped)</td><td colspan="4"></td></tr>
<tr><td>Rotation Y</td><td></td><td>0. ° (ramped)</td><td colspan="4"></td></tr>
<tr><td>Rotation Z</td><td></td><td>0. ° (ramped)</td><td colspan="4"></td></tr>
<tr><td>Behavior</td><td></td><td colspan="2">Deformable</td><td colspan="3"></td></tr>
<tr><td>Rotation X</td><td colspan="2"></td><td>0.1 ° (ramped)</td><td colspan="3"></td></tr>
<tr><td>Rotation Y</td><td colspan="2"></td><td>0. ° (ramped)</td><td colspan="3"></td></tr>
<tr><td>Rotation Z</td><td colspan="2"></td><td>0. ° (ramped)</td><td colspan="3"></td></tr>
<tr><th colspan="7">Advanced</th></tr>
<tr><td>Pinball Region</td><td></td><td colspan="2">All</td><td colspan="3"></td></tr>
</table>

Case 2 – Lateral loads – Results - Structure steel S235:

Total deformation:

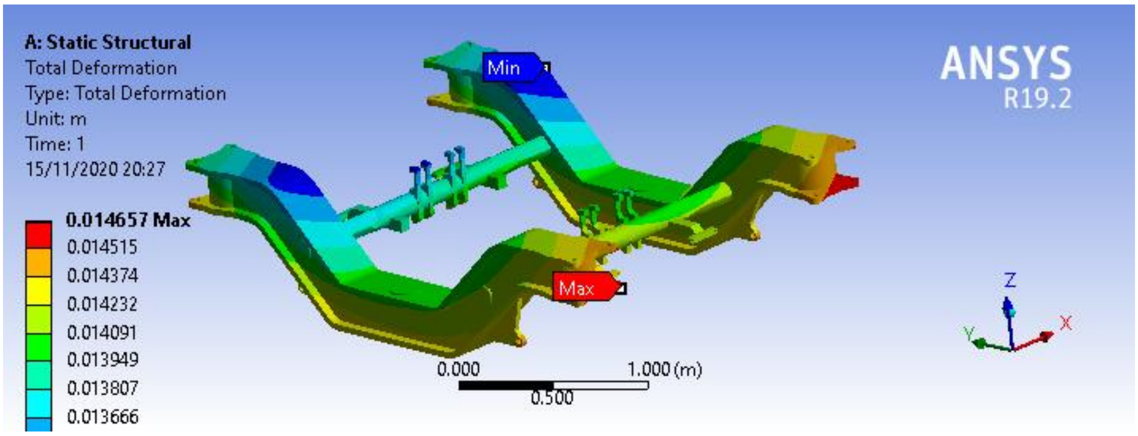


Figure 11. Total deformation 1 - structure steel - case 2

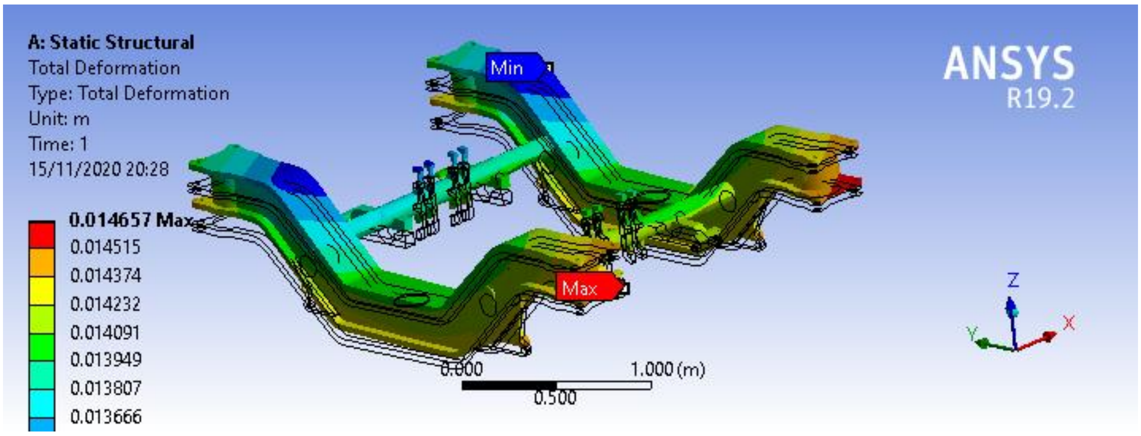


Figure 12. Total deformation 2 - structure steel - case 2

Von-Misses stress:

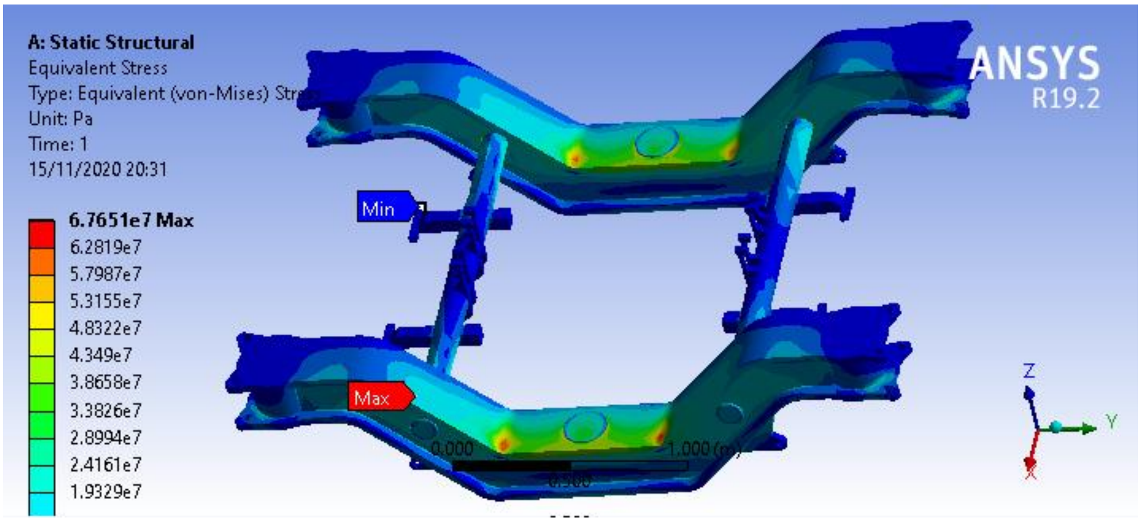


Figure 13. Von-Mises stress 1 - structure steel - case 2.

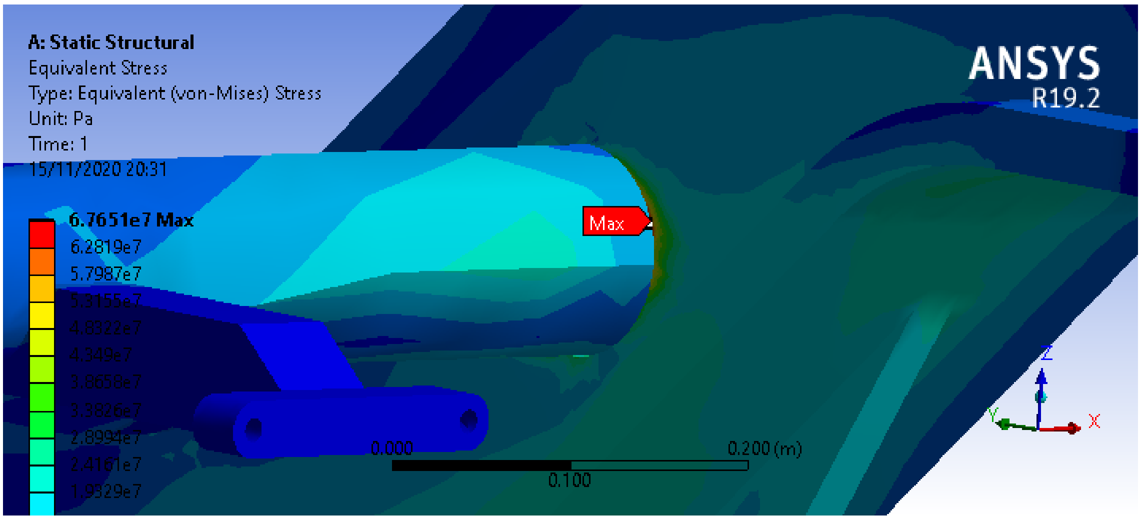


Figure 14. Von-Mises stress 2 - structure steel - case 2

Safety factor:

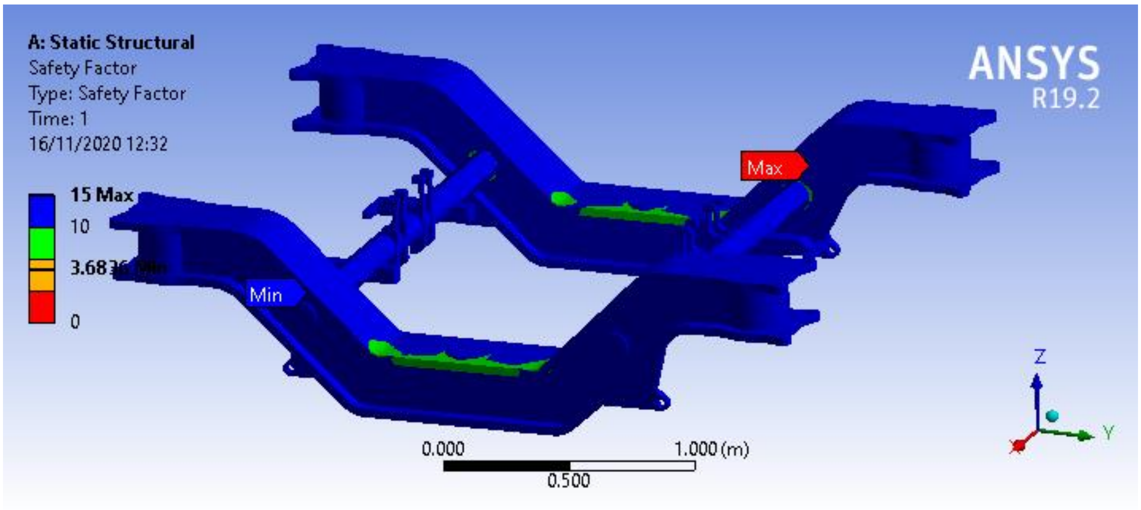


Figure 15. Safety factor 1 - structure steel - case 2.

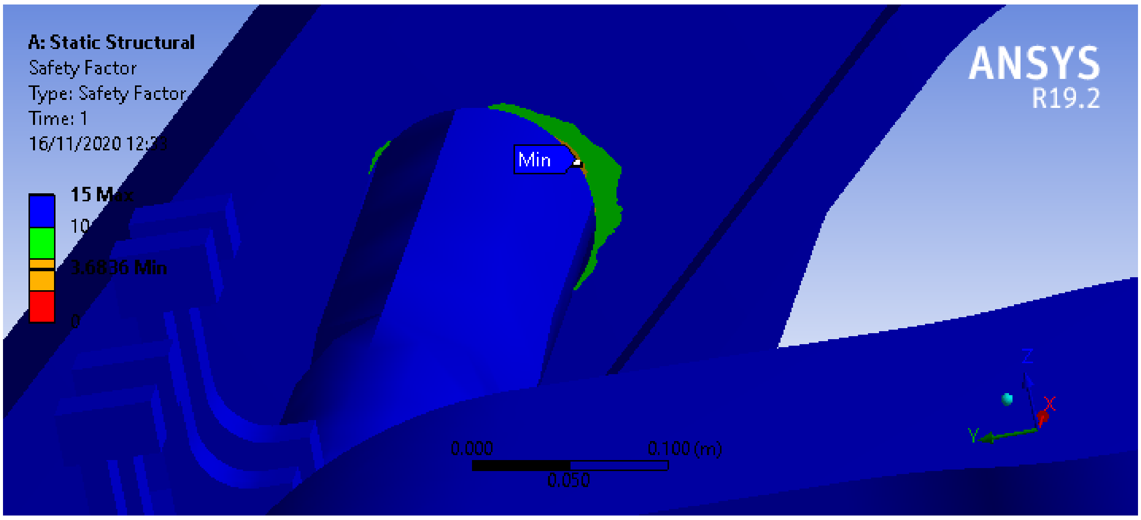


Figure 16. Safety factor 2 - structure steel - case 2

Case 3 - Unloading loads – Boundary conditions - Structure steel S235;

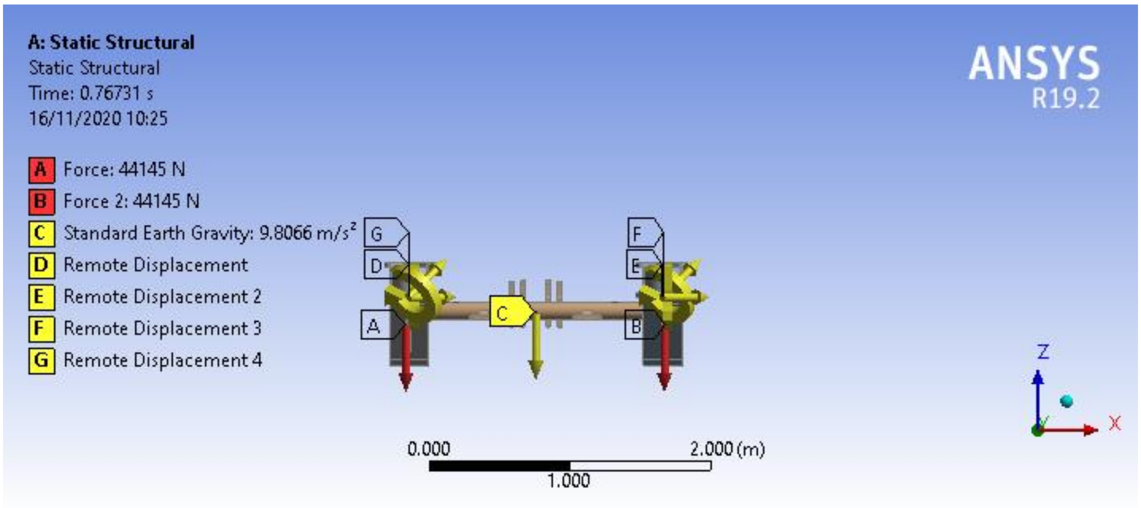


Figure 17. Boundary and unloading loads for structure steel S235, view 1.

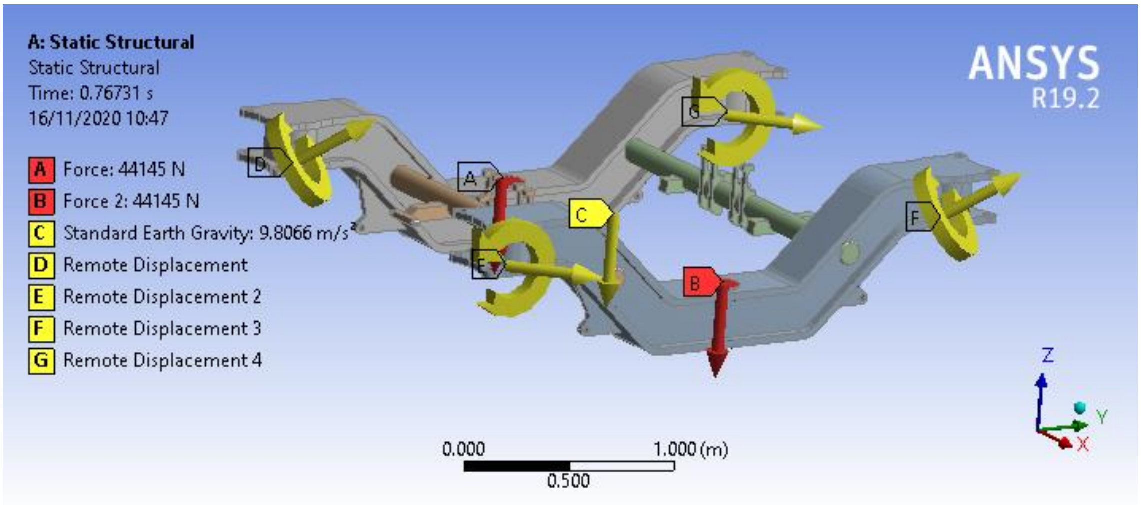


Figure 18. Boundary and unloading forces for structure steel S235, view 2.

Table 3. Detailed table of loads and restrictions for case 3 of the Structure steel S235.

Definition					
Type	Force	Remote Displacement			
Define By	Vector				
Magnitude	-44145 N (ramped)				
Direction	Defined				
Suppressed	No				
X Component		0.01 m (ramped)			
Y Component		0.01 m (ramped)			
Z Component		0.01 m (ramped)	0. m (ramped)	0.01 m (ramped)	0. m (ramped)
Rotation X		0.1 ° (ramped)			
Rotation Y		0.1 ° (ramped)			
Rotation Z		0.1 ° (ramped)			
Behavior		Deformable			
Rotation X			0.1 ° (ramped)		
Rotation Y			0. ° (ramped)		
Rotation Z			0. ° (ramped)		
Rotation X				0.1 ° (ramped)	
Rotation Y				0.1 ° (ramped)	
Rotation Z				0.1 ° (ramped)	
Rotation X					0.1 ° (ramped)
Rotation Y					0. ° (ramped)
Rotation Z					0. ° (ramped)
Advanced					
Pinball Region		All			

Case 3 - Unloading loads – Results - Structure steel S235;

Total deformation:

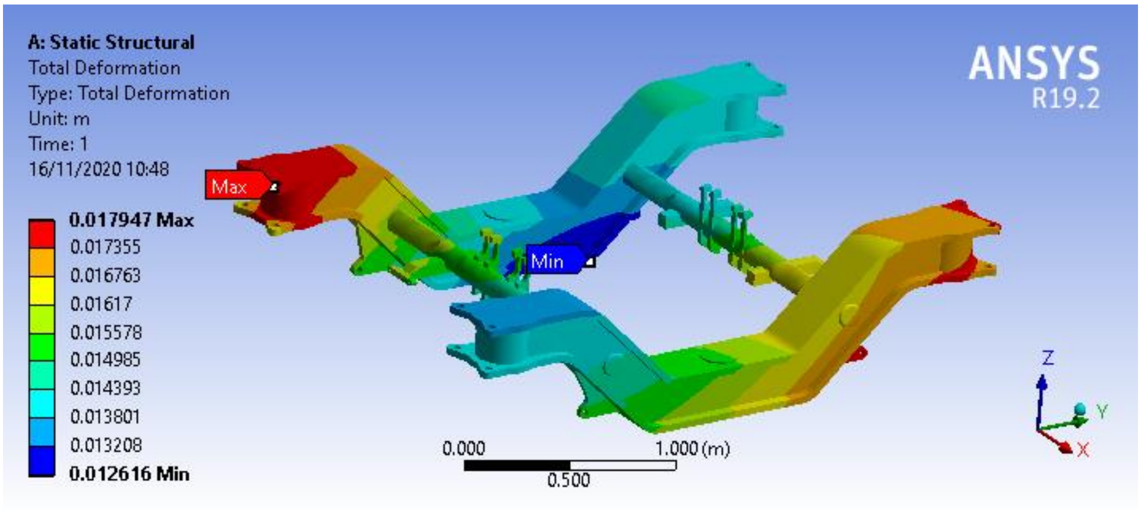


Figure 19. Total deformation - structure steel - case 3

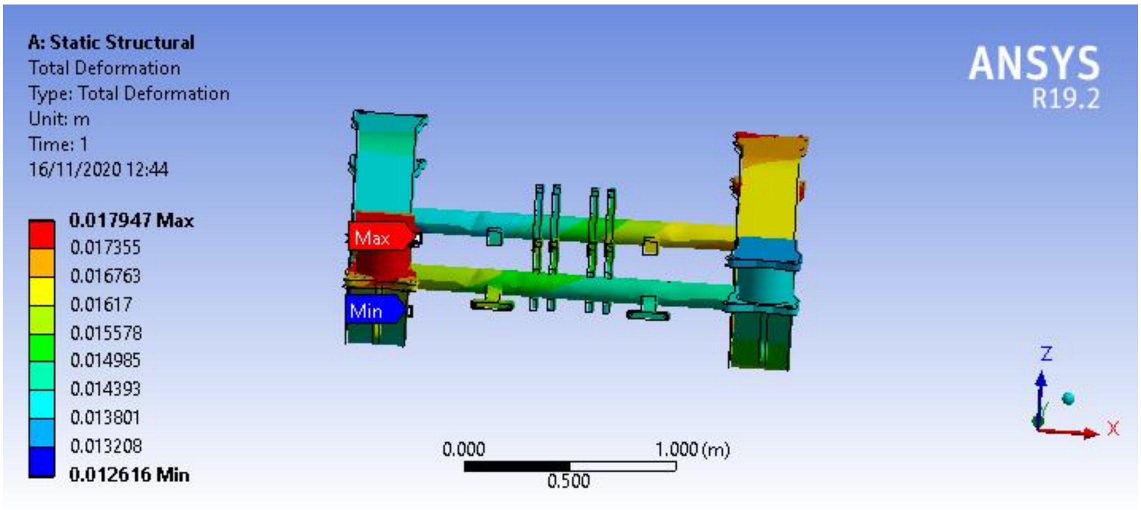


Figure 20.Total deformation 2 - structure steel - case 3.

Von-Misses stress:

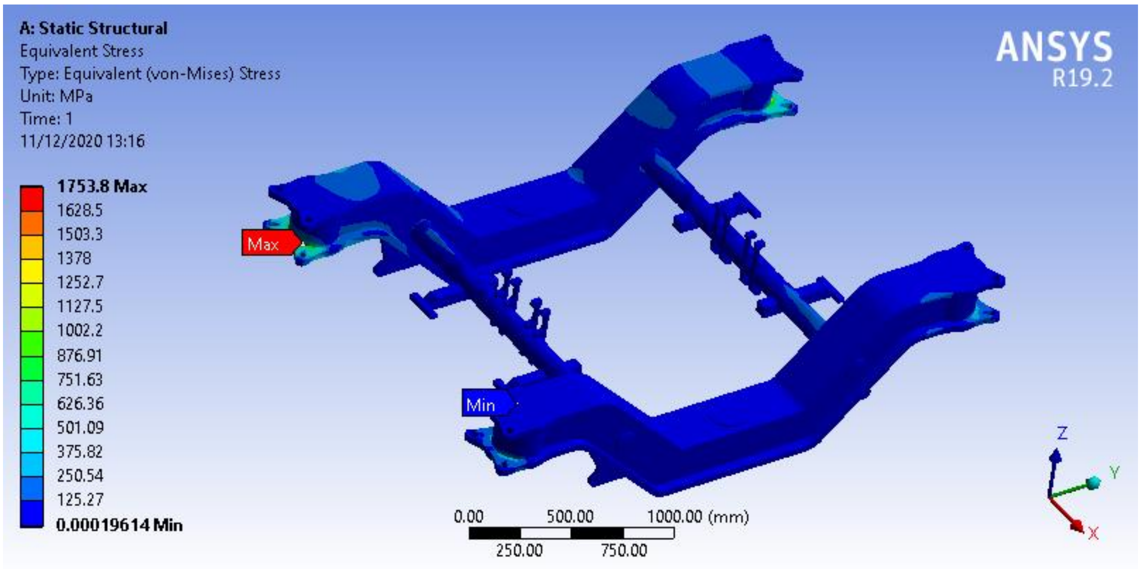


Figure 21. Von-Mises stress - structure steel - case 3.

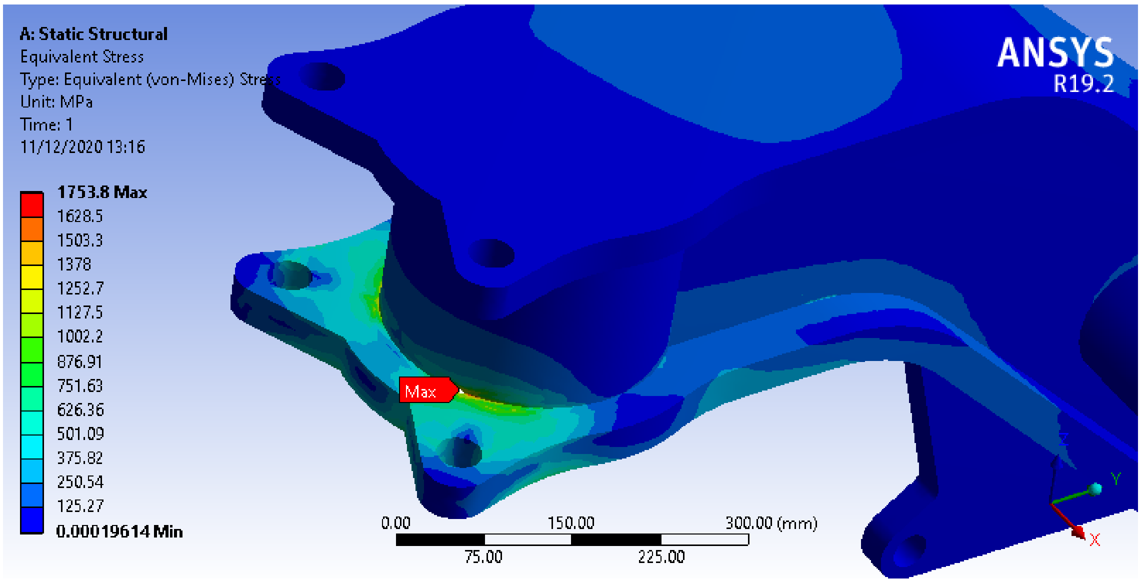


Figure 22. Von-Mises stress 2 - structure steel - case 3.

Safety factor:

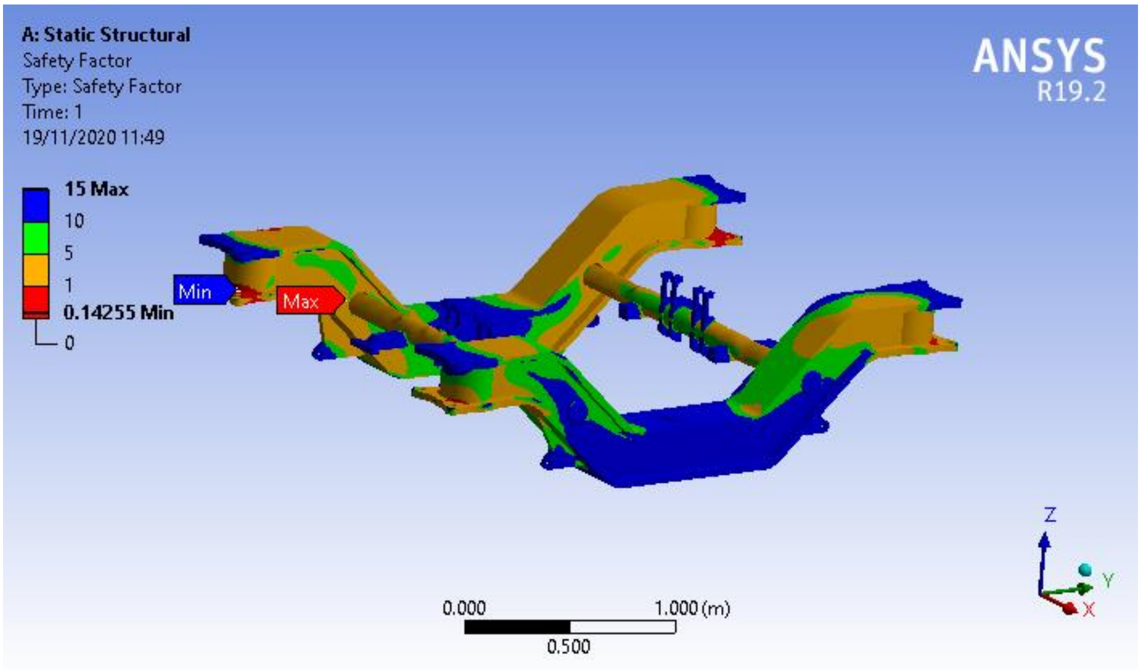


Figure 23. Safety factor - structure steel - case 3.

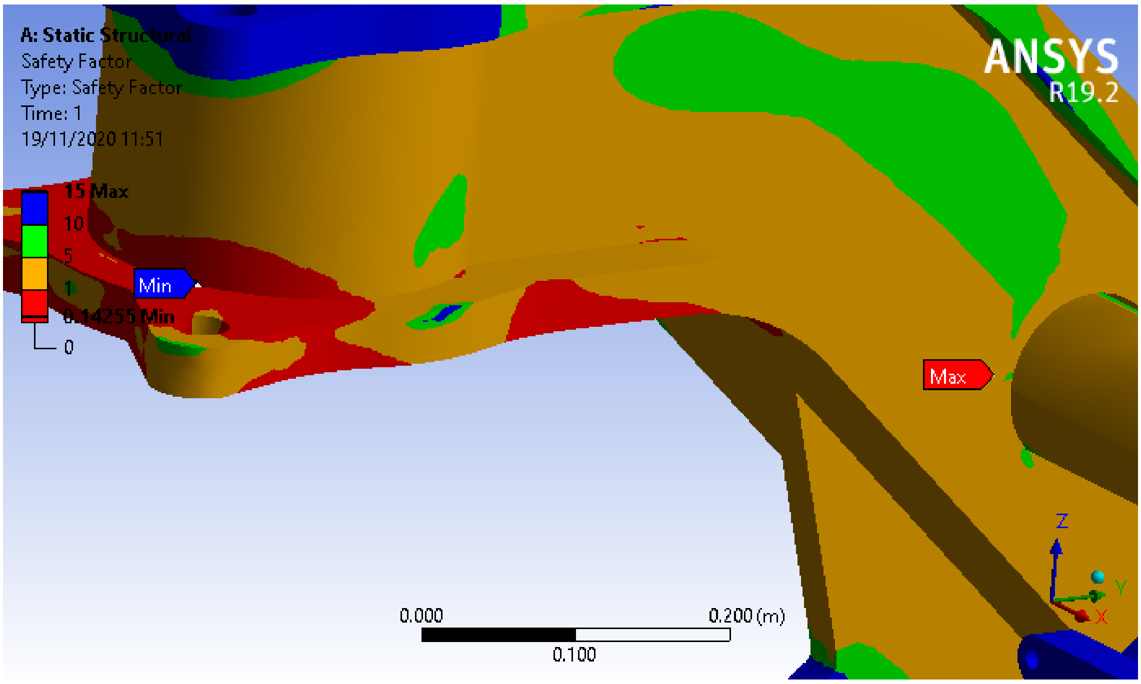


Figure 24. Safety factor 2 - structure steel - case 3.

Case 4 - Lonzening loads – Boundary conditions - Structure steel S235;

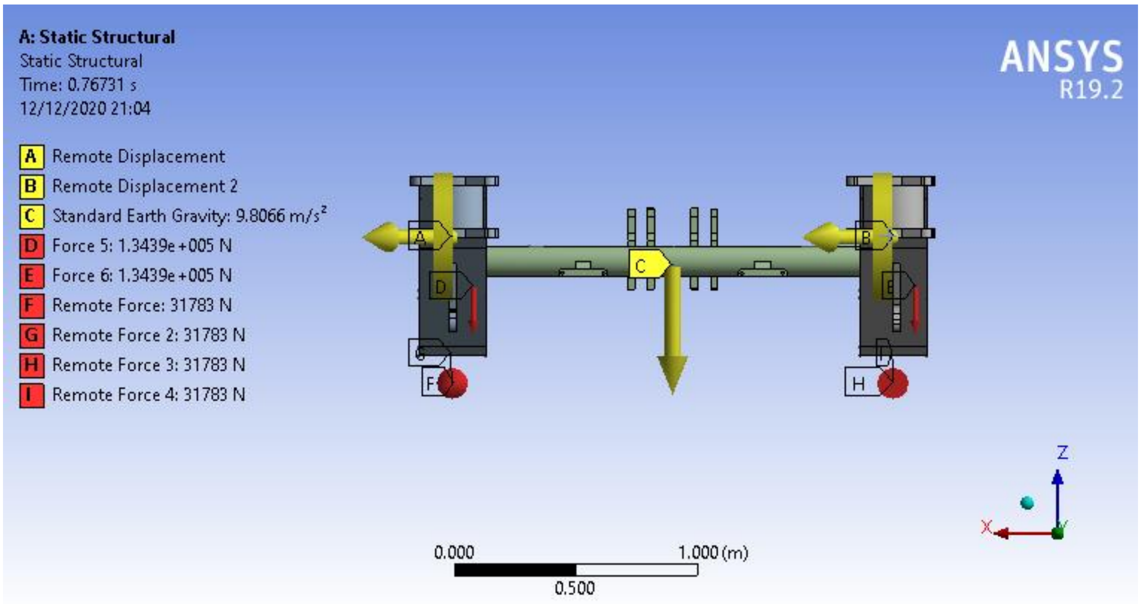


Figure 25. Boundary and lozenging loads for structure steel S235, view 1.

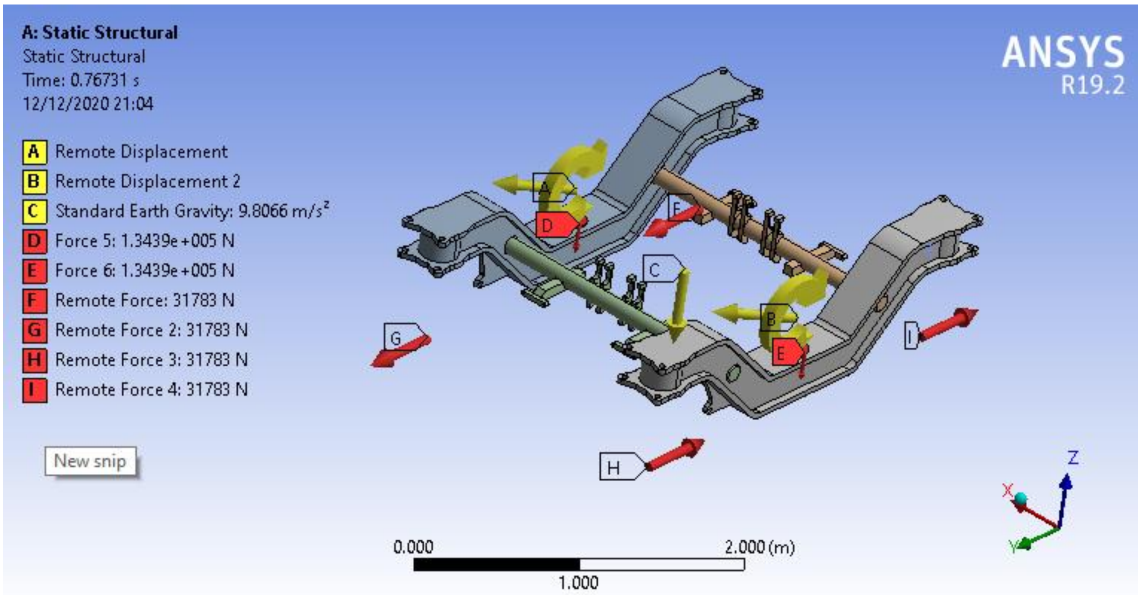


Figure 26. Boundary and lozenging loads for structure steel S235, view 2.

Table 4. Detailed table of loads and restrictions for case 4 of the Structure steel S235.

Definition								
Object Name	*Remote Displacement*	*Remote Displacement 2*	*Force*	*Force 2*	*Force 3*	*Force 4*	*Force 5*	*Force 6*
Type	Remote Displacement		Force					
X Component	0.01 m (ramped)		0. N (ramped)					
Y Component	0.01 m (ramped)		31783 N (ramped)		-31783 N (ramped)	0. N (ramped)		
Z Component	0. m (ramped)		0. N (ramped)			-134388 N (ramped)		
Rotation X	0.1 ° (ramped)							
Rotation Y	0. ° (ramped)							
Rotation Z	0. ° (ramped)							
Suppressed	No							
Behavior	Deformable							
Rotation X		0.1 ° (ramped)						
Rotation Y		0. ° (ramped)						
Rotation Z		0. ° (ramped)						
Define By			Components					
Coordinate System			Global Coordinate System					
Advanced								
Pinball Region	All							

Case 4 - Lonzening loads – Results - Structure steel S235;

Total deformation:

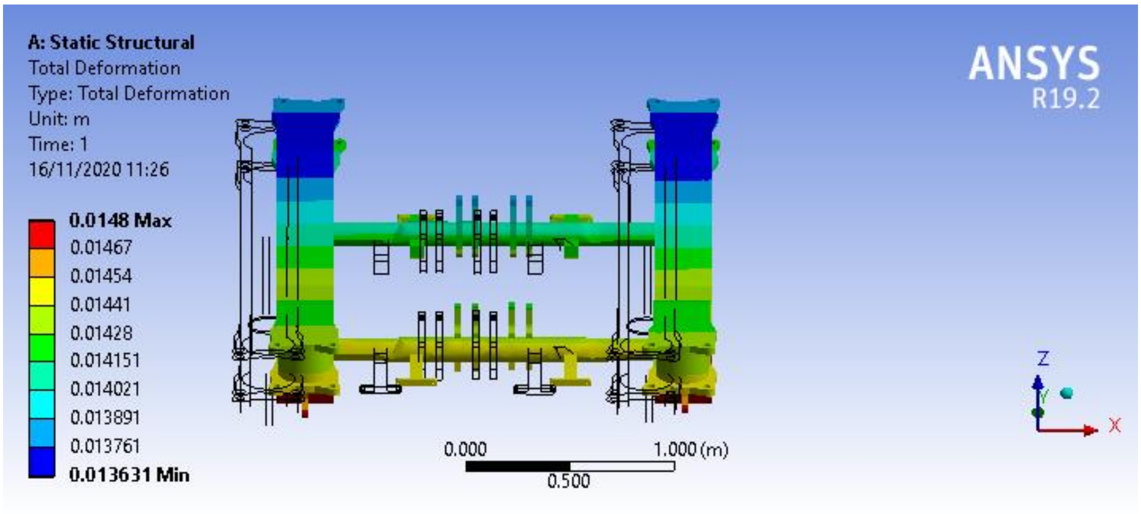


Figure 27. total deformation 1 - structure steel - case 4

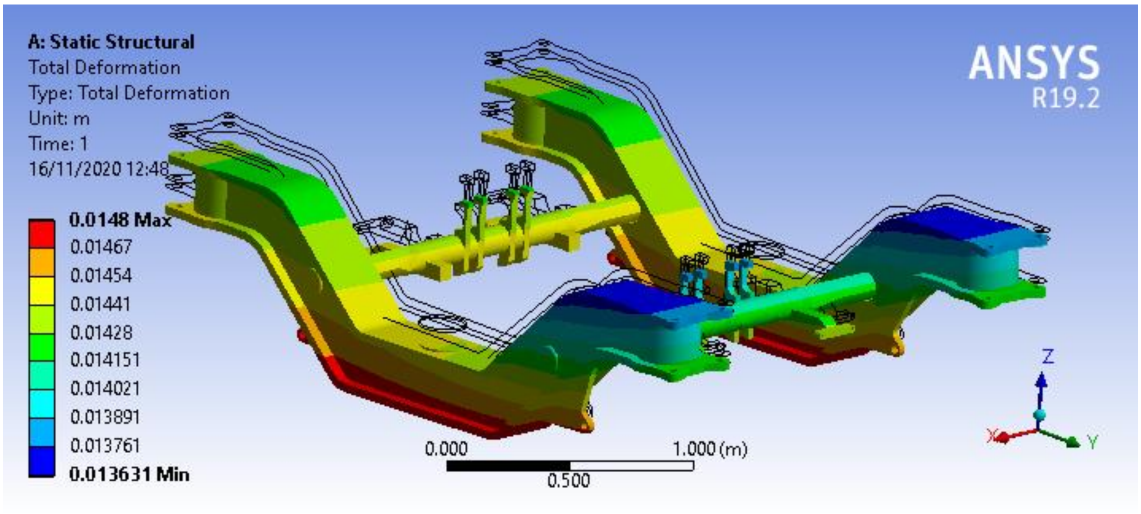


Figure 28. total deformation 2 - structure steel - case 4

UNIVERSITAT POLITÈCNICA DE CATALUNYA
BARCELONATECH
Escola d'Enginyeria de Barcelona Est

Von-mises stress

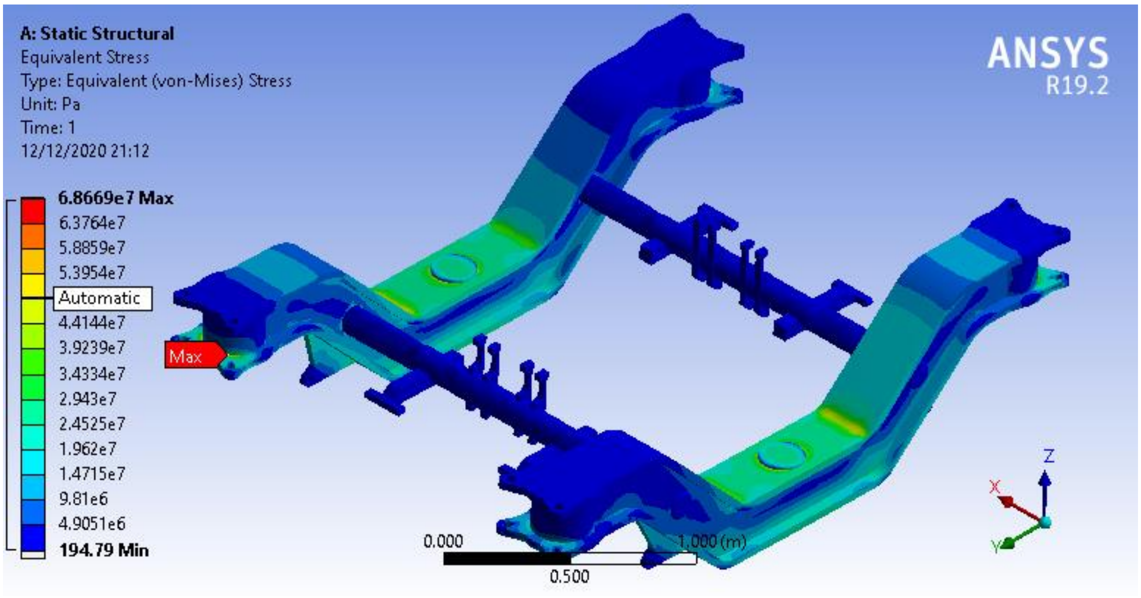


Figure 29. Von-Mises stress 1 - structure steel - case 4.

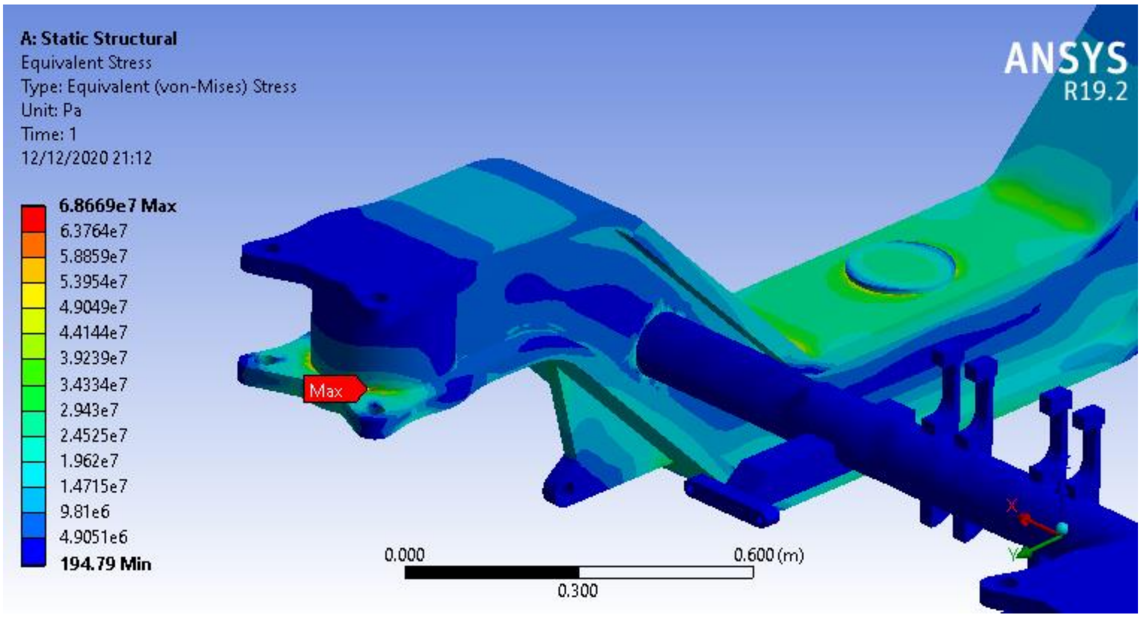


Figure 30. Von-Mises stress 2 - structure steel - case 4

Safety factor:

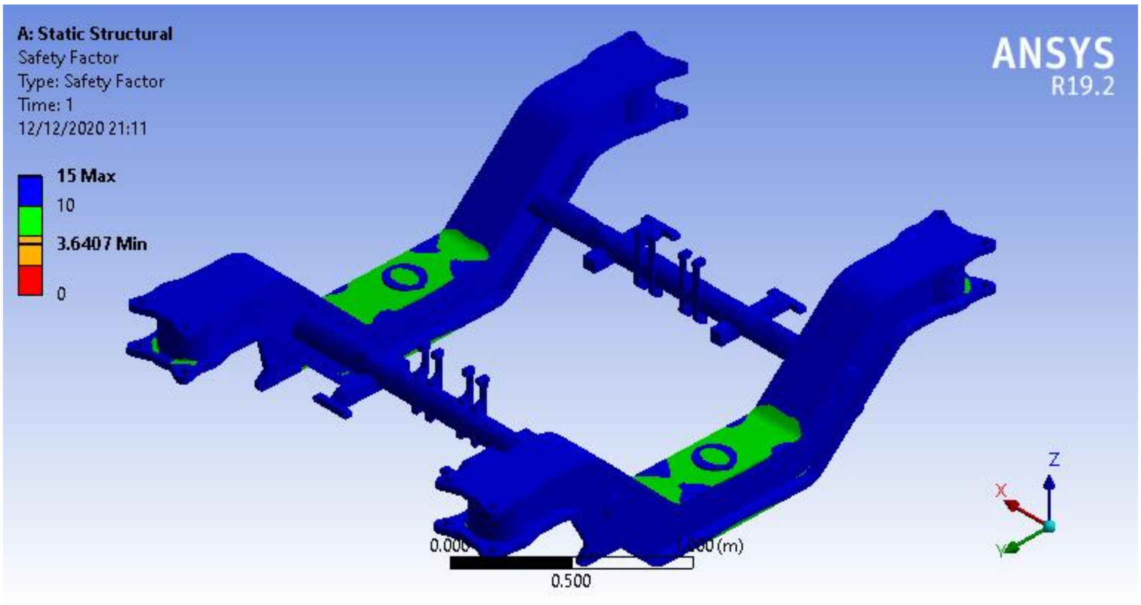


Figure 31. Safety factor 1 - structure steel - case 4

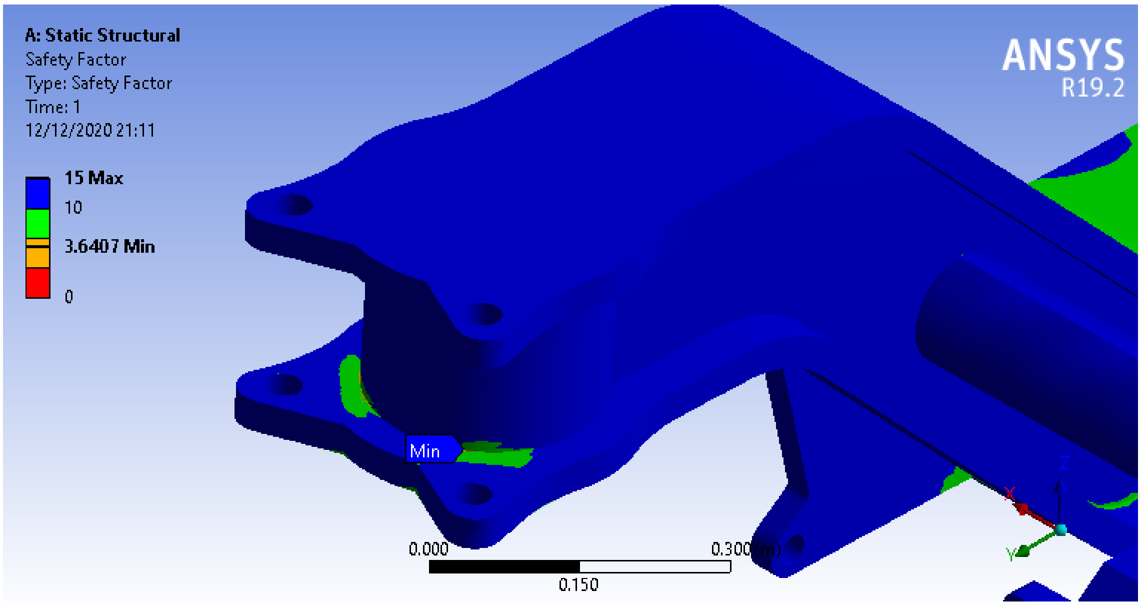


Figure 32. Safety factor 2 - structure steel - case 4

Case 5 - Longitudinal loads – Boundary conditions - Structure steel S235;

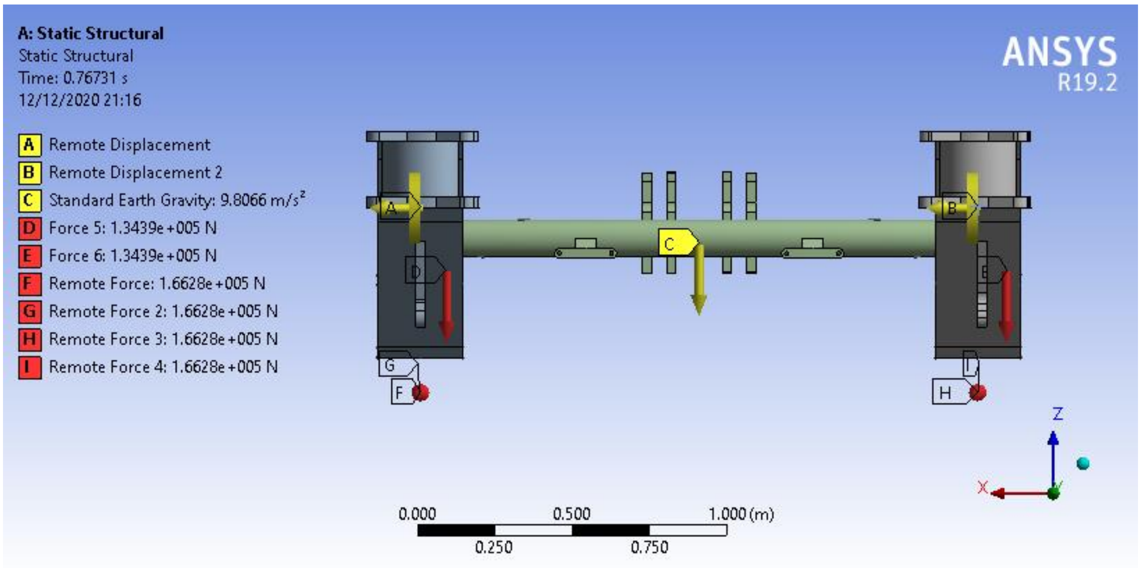


Figure 33. Boundary and longitudinal shunting loads for structure steel S235, view 1.

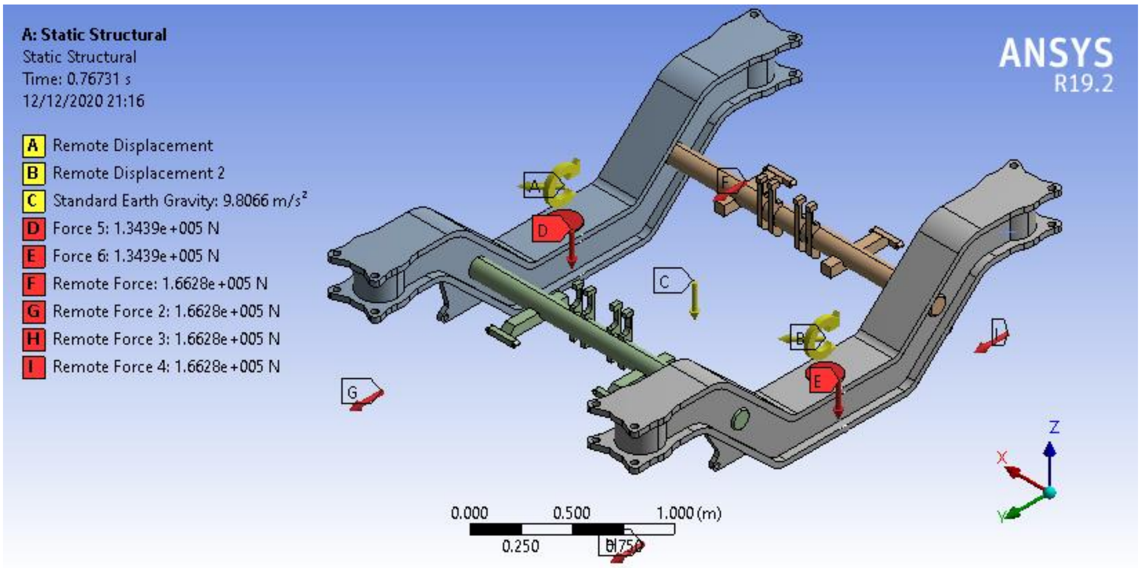


Figure 34. Boundary and longitudinal shunting loads for structure steel S235, view 2.

Table 5. Detailed table of loads and restrictions for case 5 of the Structure steel S235.

Definition								
Object Name	*Remote Displacement*	*Remote Displacement 2*	*Force 1*	*Force 2*	*Force 3*	*Force 4*	*Force 5*	*Force 6*
Type	Remote Displacement		Force					
X Component	0.01 m (ramped)		0. N (ramped)					
Y Component	0.01 m (ramped)		16628 N (ramped)				0. N (ramped)	
Z Component	0. m (ramped)		0. N (ramped)				-134388 N (ramped)	
Rotation X	0.1 ° (ramped)							
Rotation Y	0. ° (ramped)							
Rotation Z	0. ° (ramped)							
Suppressed	No							
Behavior	Deformable							
Rotation X		0.1 ° (ramped)						
Rotation Y		0. ° (ramped)						
Rotation Z		0. ° (ramped)						
Define By			Components					
Coordinate System			Global Coordinate System					
Advanced								
Pinball Region	All							

Case 5 - Longitudinal loads – Results - Structure steel S235;

Total deformation:

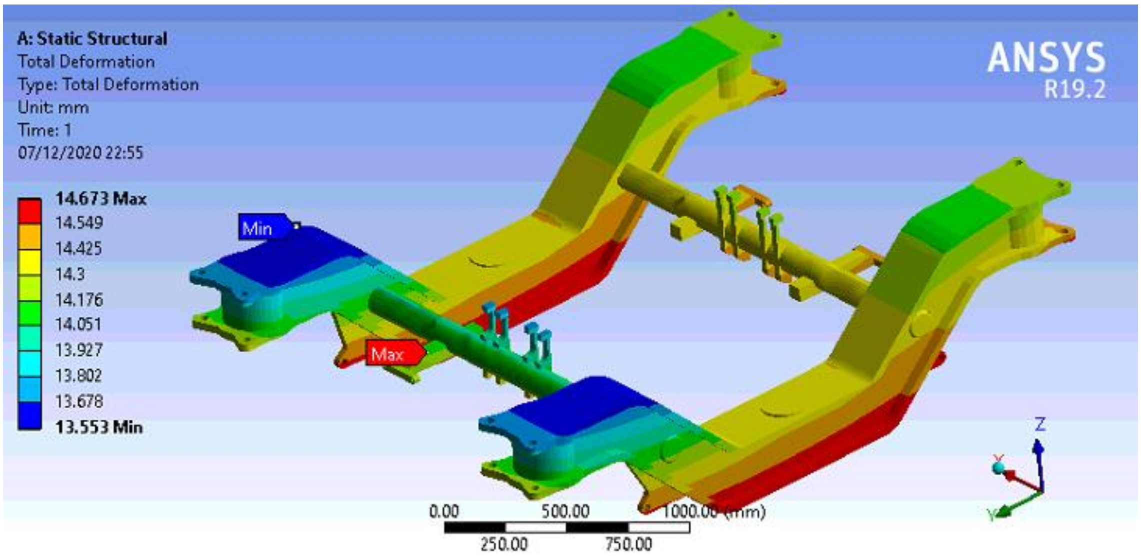


Figure 35. Total deformation 1 - structure steel - case 5

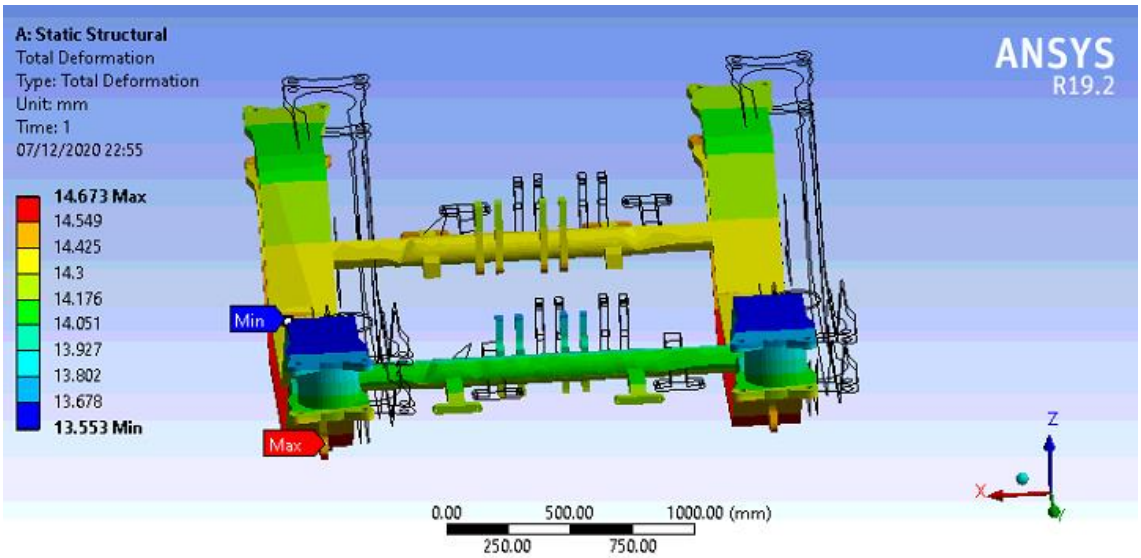


Figure 36. Total deformation 2 - structure steel - case 5

Von-Misses stress:

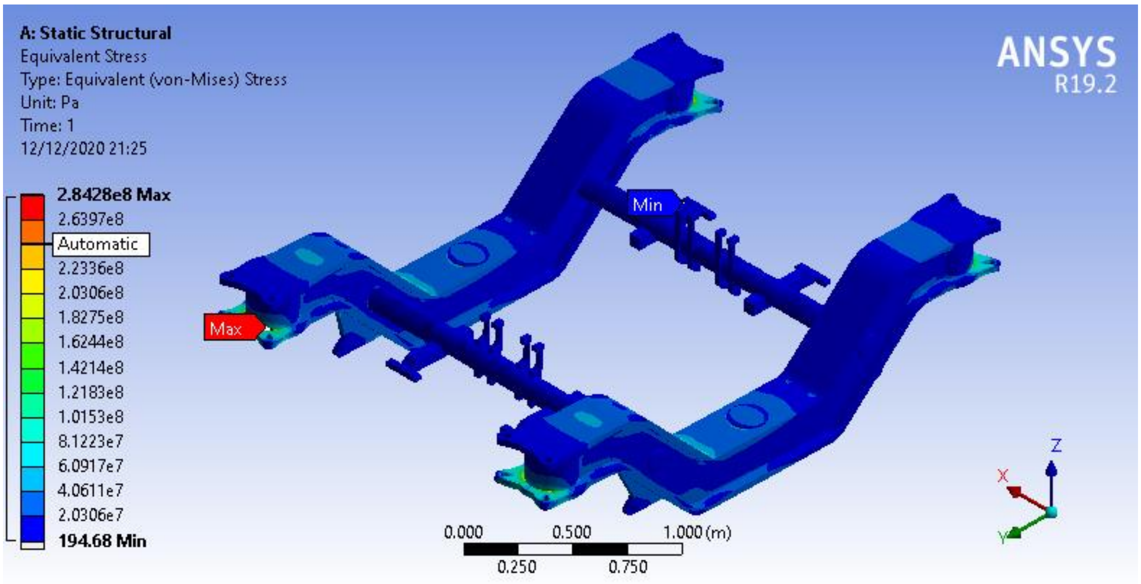


Figure 37. Von-mises stress 1 - structure steel - case 5

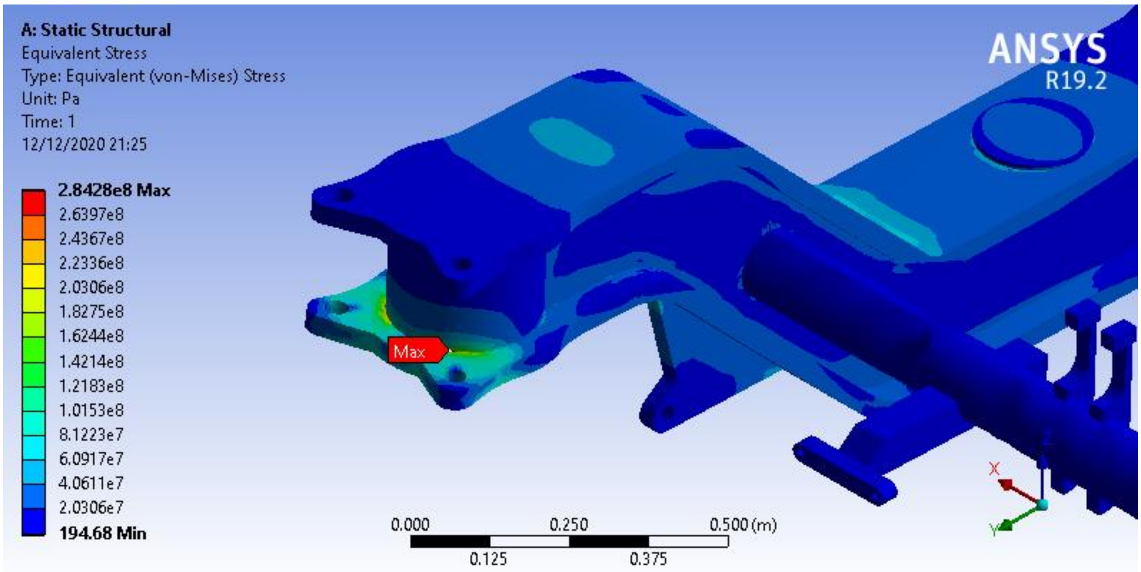


Figure 38. Von-Mises stress 2 - structure steel - case 5

Safety factor:

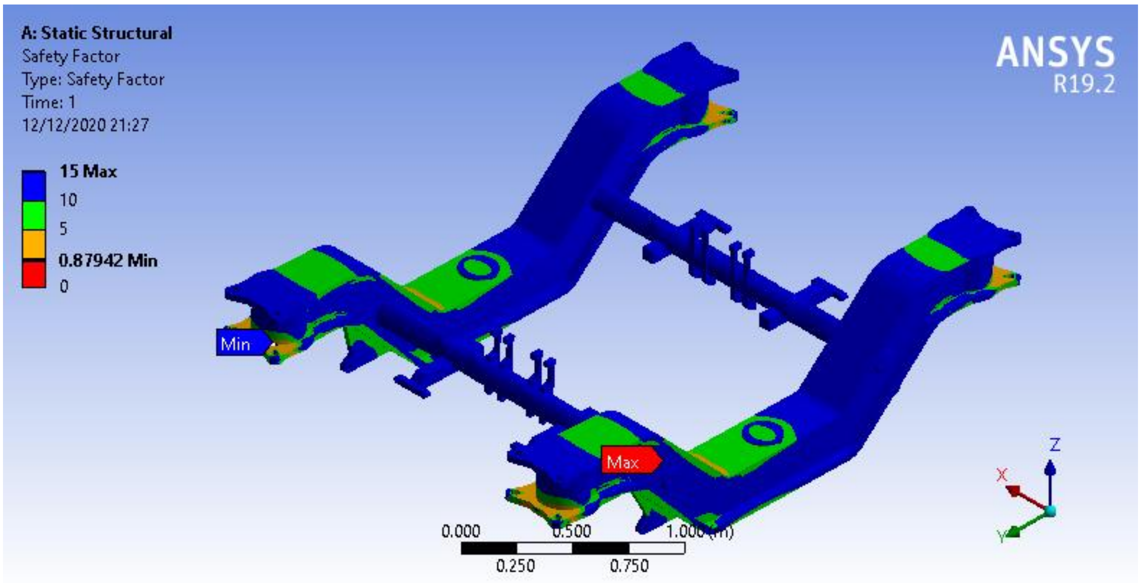


Figure 39. Safety factor 1 - structure steel - case 5

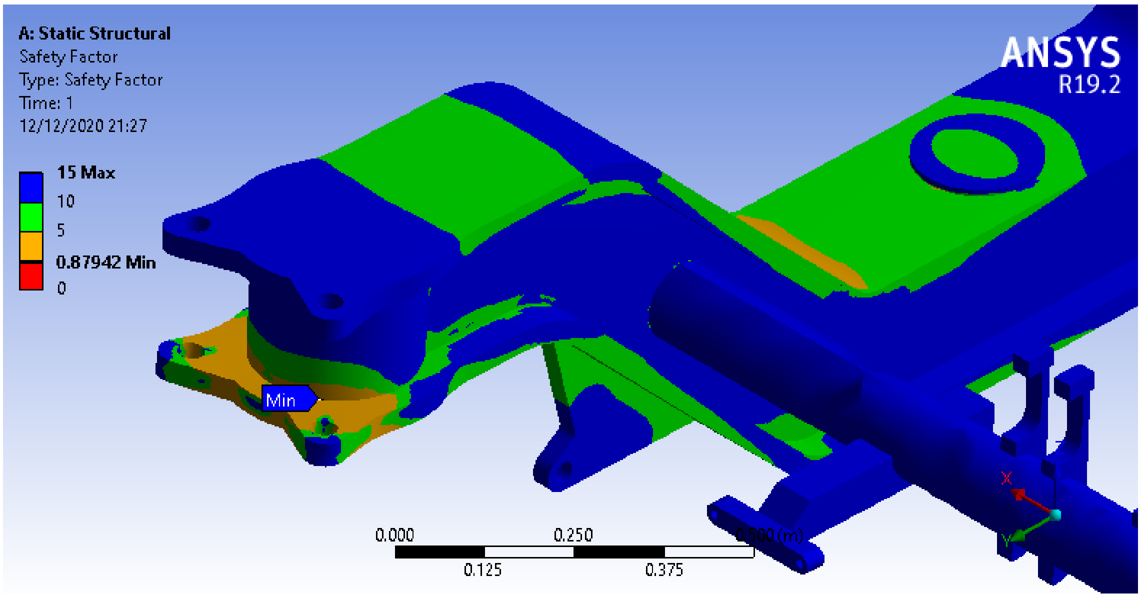


Figure 40. Safety factor 2 - structure steel - case 5

Case 6 - Fatigue vertical loads – Boundary conditions - Structure steel S235;

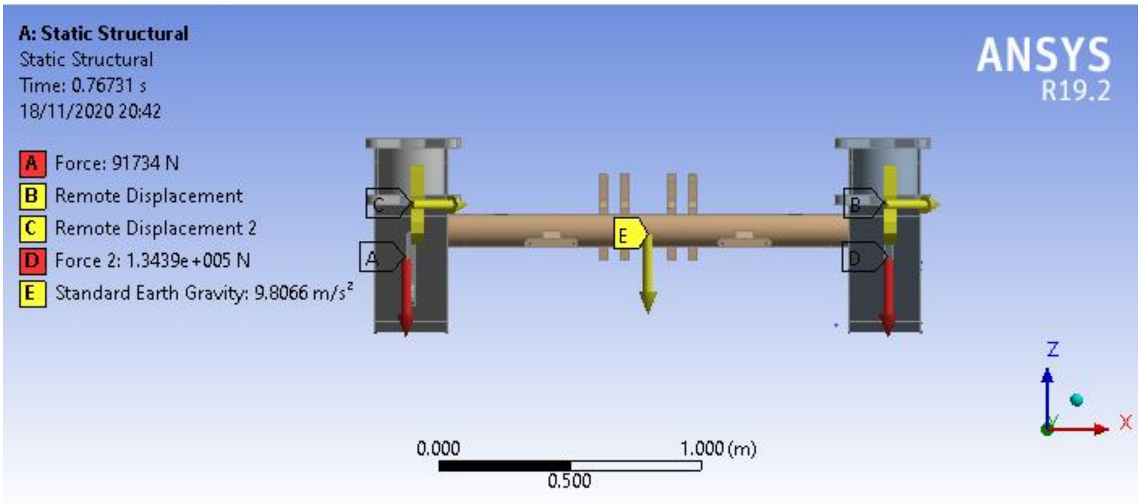


Figure 41. Boundary and vertical fatigue load for structure steel S235, view 1.

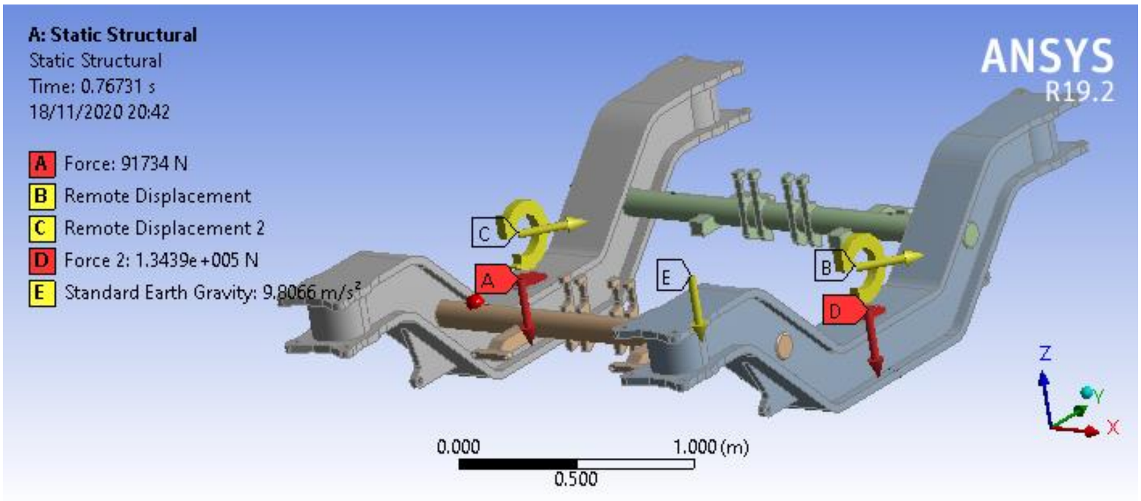


Figure 42. Boundary and vertical fatigue load for structure steel S235, view 2.

Table 6. Detailed table of loads and restrictions for case 6 of the Structure steel S235.

Object Name	*Force*	*Remote Displacement*	*Remote Displacement 2*	*Force 2*
Definition				
Type	Force	Remote Displacement		Force
Define By	Vector			Vector
Magnitude	-91734 N (ramped)			-91734 N (ramped)
Direction	Defined			Defined
Suppressed	No			
X Component		0.01 m (ramped)		
Y Component		0.01 m (ramped)		
Z Component		0. m (ramped)		
Rotation X		0.1 ° (ramped)		
Rotation Y		0. ° (ramped)		
Rotation Z		0. ° (ramped)		
Behavior		Deformable	Coupled	
Rotation X			0.1 ° (ramped)	
Rotation Y			0. ° (ramped)	
Rotation Z			0. ° (ramped)	
Advanced				
Pinball Region		All		

Case 6 - Vertical fatigue loads – Results - Structure steel S235;

Safety factor:

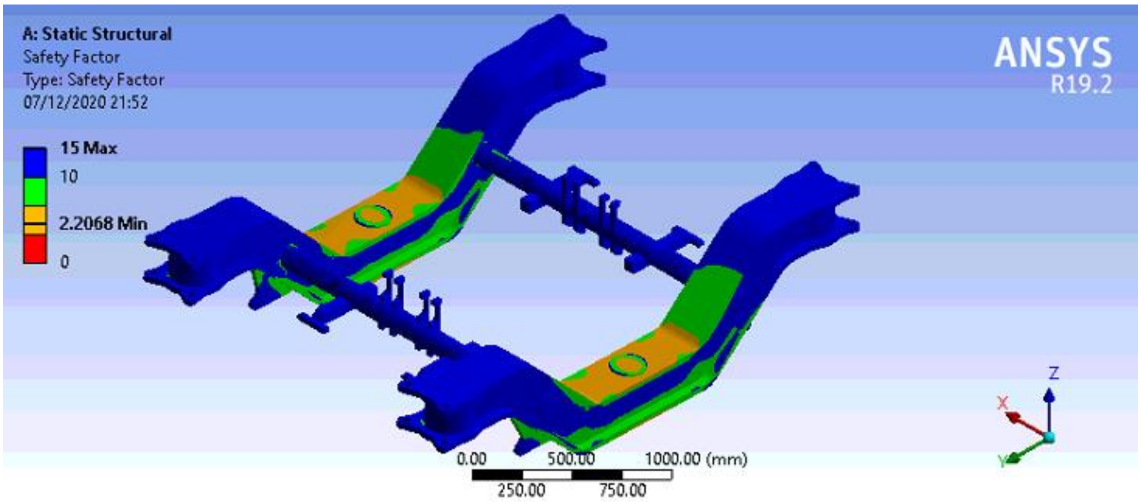


Figure 43. Safety factor (fatigue) - structure steel - case 6

Life expentancy:

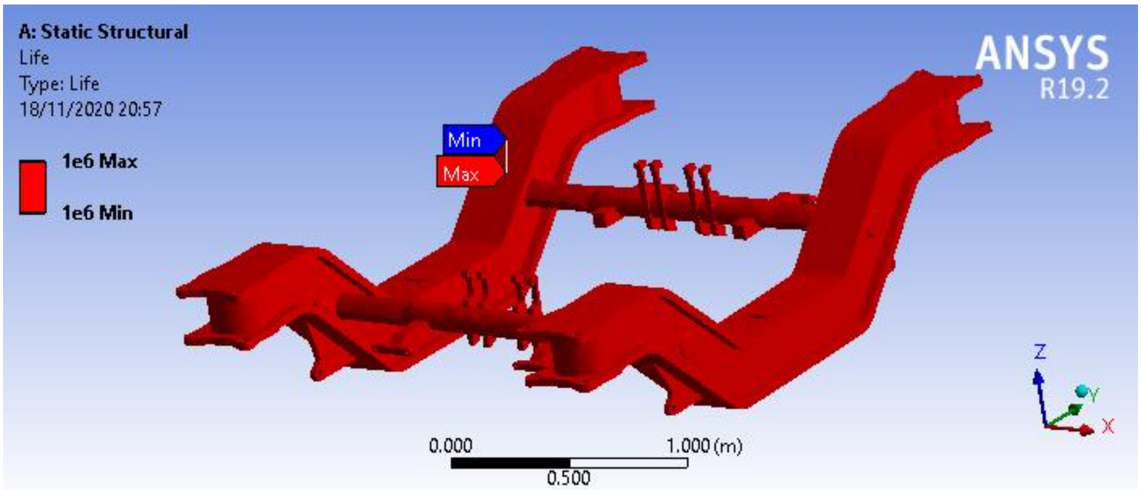


Figure 44. Life expectancy (fatigue) - structure steel - case 6

Case 7 - Fatigue Lateral loads – Boundary conditions - Structure steel S235;

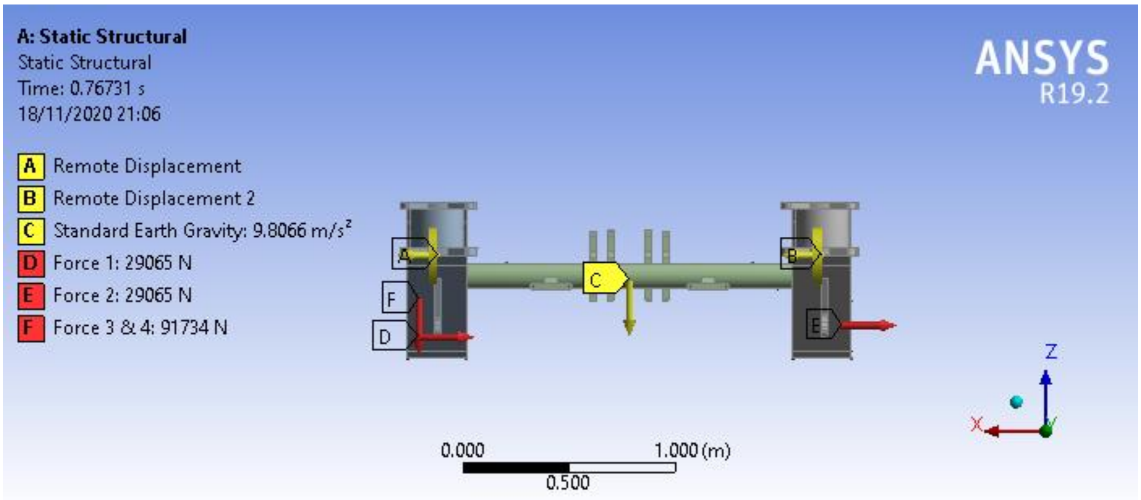


Figure 45. Boundary and lateral fatigue load for structure steel S235, view 1.

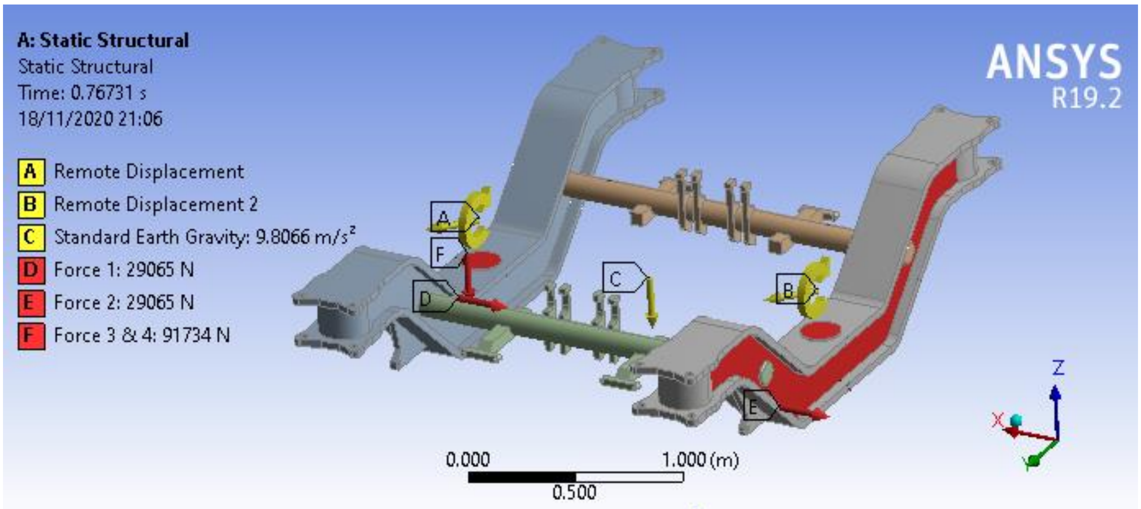


Figure 46. Boundary and lateral fatigue load for structure steel S235 s, view 2.

Table 7. Detailed table of loads and restrictions for case 7 of the Structure steel S235.

Object Name	*Remote Displacement*	*Remote Displacement 2*	*Force 1*	*Force 2*	*Force 3 & 4*
Definition					
Type	Remote Displacement		Force		
X Component	0.01 m (ramped)				
Y Component	0.01 m (ramped)				
Z Component	0. m (ramped)				
Rotation X	0.1 ° (ramped)				
Rotation Y	0. ° (ramped)				
Rotation Z	0. ° (ramped)				
Suppressed	No				
Behavior	Deformable				
Rotation X		0.1 ° (ramped)			
Rotation Y		0. ° (ramped)			
Rotation Z		0. ° (ramped)			
Define By			Vector		
Magnitude			-29065 N (ramped)		-91734 N (ramped)
Direction			Defined		
Advanced					
Pinball Region	All				

Case 7 - Lateral fatigue loads – Results - Structure steel S235;

Safety factor:

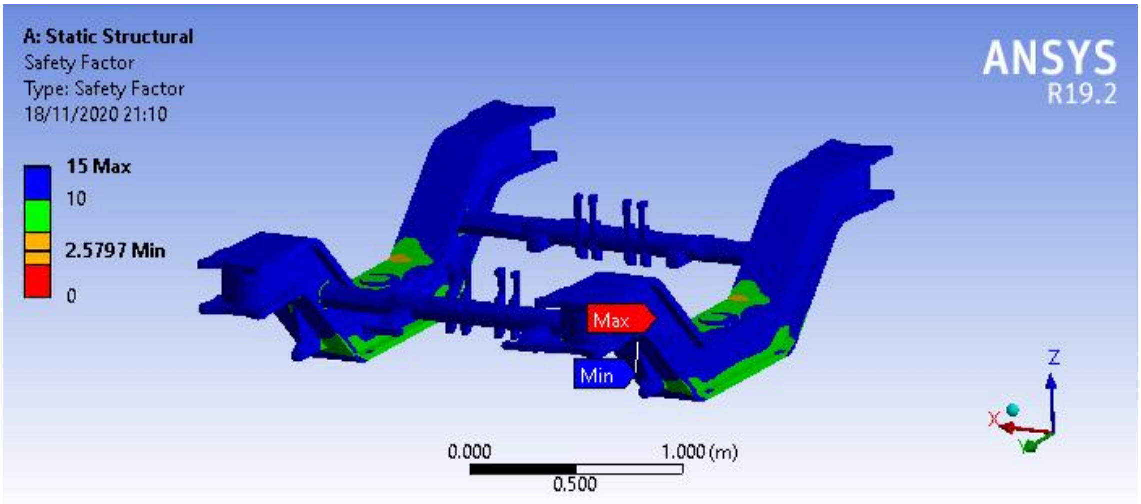


Figure 47. Safety factor (fatigue) - structure steel - case 7

Life expectancy:

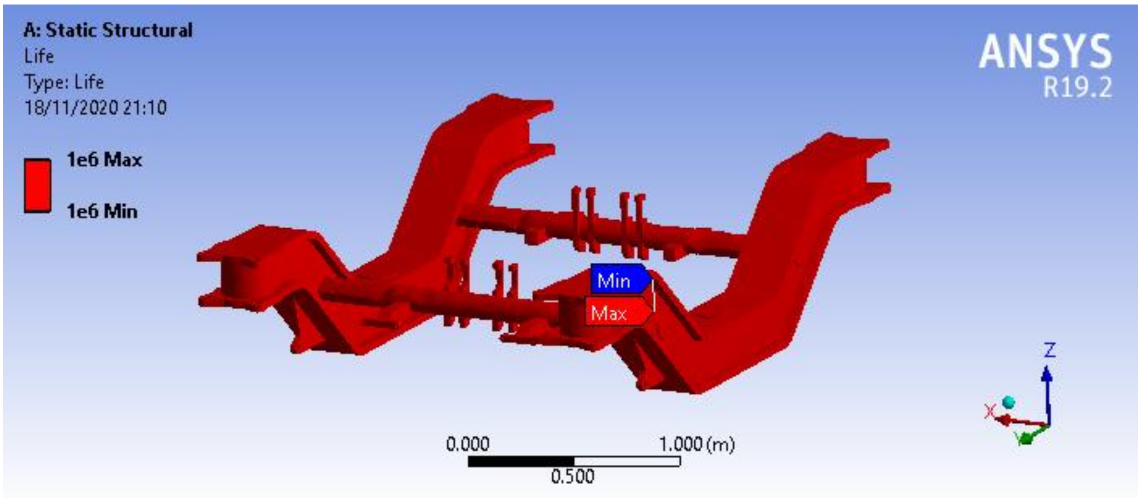


Figure 48. Life expectancy (fatigue) - structure steel - case 7

Case 8 - Fatigue Lonzenging loads – Boundary conditions - Structure steel S235;

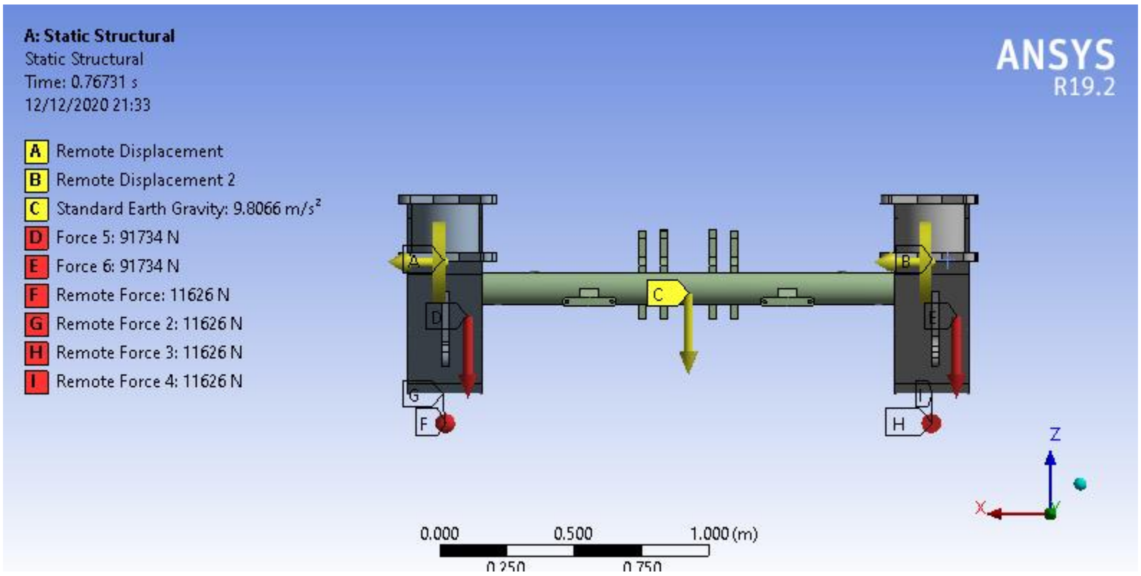


Figure 49. Boundary and lozenging fatigue load for structure steel S235, view 1.

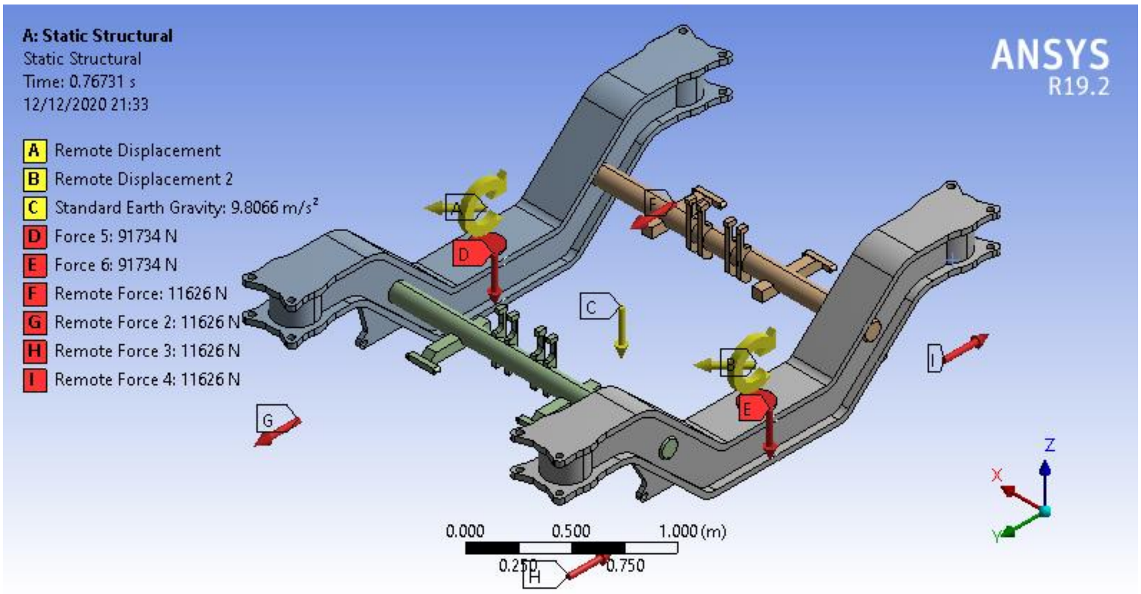


Figure 50. Boundary and lozenging fatigue load for structure steel S235, view 2.

Table 8. Detailed table of loads and restrictions for case 8 of the Structure steel S235.

<table>
<tr><th>Object Name</th><th>Remote Displacement</th><th>Remote Displacement 2</th><th>Force</th><th>Force 2</th><th>Force 3</th><th>Force 4</th><th>Force 5</th><th>Force 6</th></tr>
<tr><td colspan="9">Definition</td></tr>
<tr><td>Type</td><td colspan="2">Remote Displacement</td><td colspan="6">Force</td></tr>
<tr><td>X Component</td><td colspan="2">0.01 m (ramped)</td><td colspan="6">0. N (ramped)</td></tr>
<tr><td>Y Component</td><td colspan="2">0.01 m (ramped)</td><td colspan="2">11626 N (ramped)</td><td colspan="2">-11626 N (ramped)</td><td colspan="2">0. N (ramped)</td></tr>
<tr><td>Z Component</td><td colspan="2">0. m (ramped)</td><td colspan="4">0. N (ramped)</td><td colspan="2">-91734 N (ramped)</td></tr>
<tr><td>Rotation X</td><td>0.1 ° (ramped)</td><td colspan="7"></td></tr>
<tr><td>Rotation Y</td><td>0. ° (ramped)</td><td colspan="7"></td></tr>
<tr><td>Rotation Z</td><td>0. ° (ramped)</td><td colspan="7"></td></tr>
<tr><td>Suppressed</td><td colspan="8">No</td></tr>
<tr><td>Behavior</td><td colspan="2">Deformable</td><td colspan="6"></td></tr>
<tr><td>Rotation X</td><td></td><td>0.1 ° (ramped)</td><td colspan="6"></td></tr>
<tr><td>Rotation Y</td><td></td><td>0. ° (ramped)</td><td colspan="6"></td></tr>
<tr><td>Rotation Z</td><td></td><td>0. ° (ramped)</td><td colspan="6"></td></tr>
<tr><td>Define By</td><td colspan="2"></td><td colspan="6">Components</td></tr>
<tr><td>Coordinate System</td><td colspan="2"></td><td colspan="6">Global Coordinate System</td></tr>
<tr><td colspan="9">Advanced</td></tr>
<tr><td>Pinball Region</td><td colspan="2">All</td><td colspan="6"></td></tr>
</table>

Case 8 - Lonzenning fatigue loads – Results - Structure steel S235;

Safety factor:

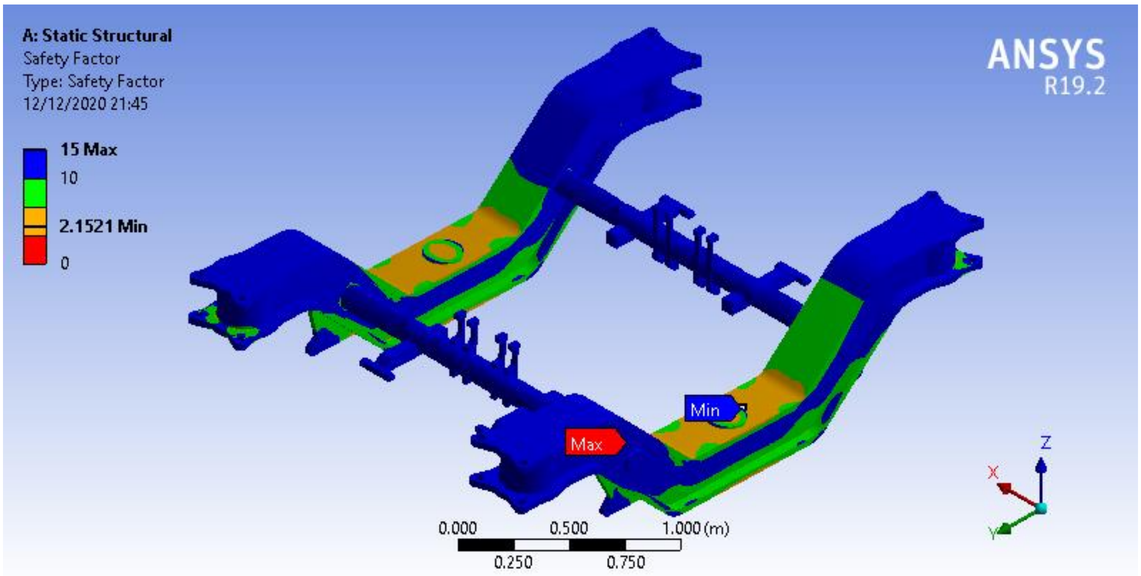


Figure 51. Safety factor (fatigue) - structure steel - case 8

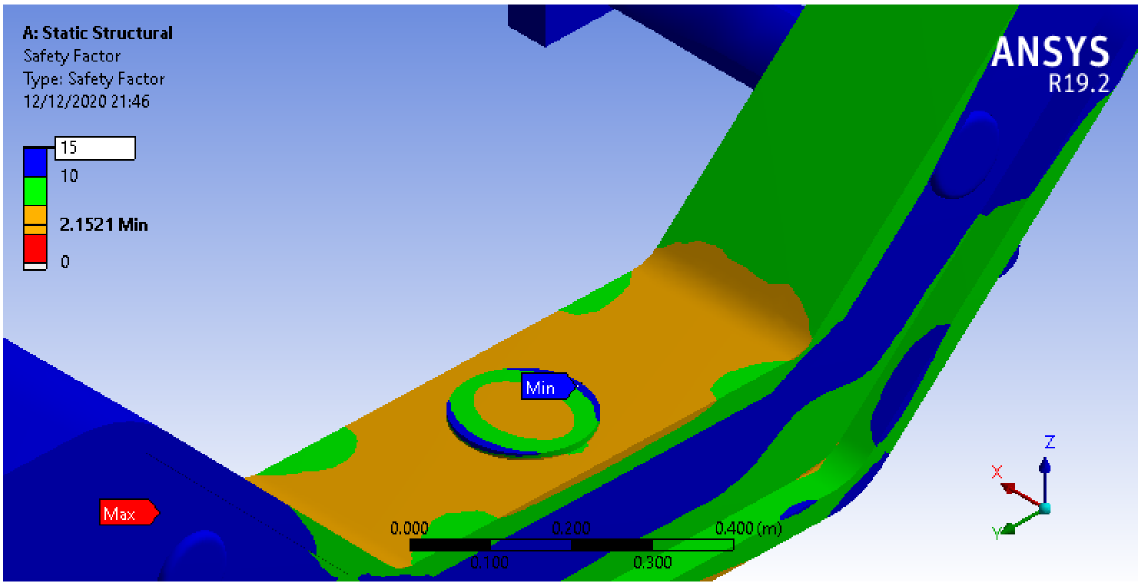


Figure 52. Safety factor (fatigue) - structure steel - case 8

Life expectancy:

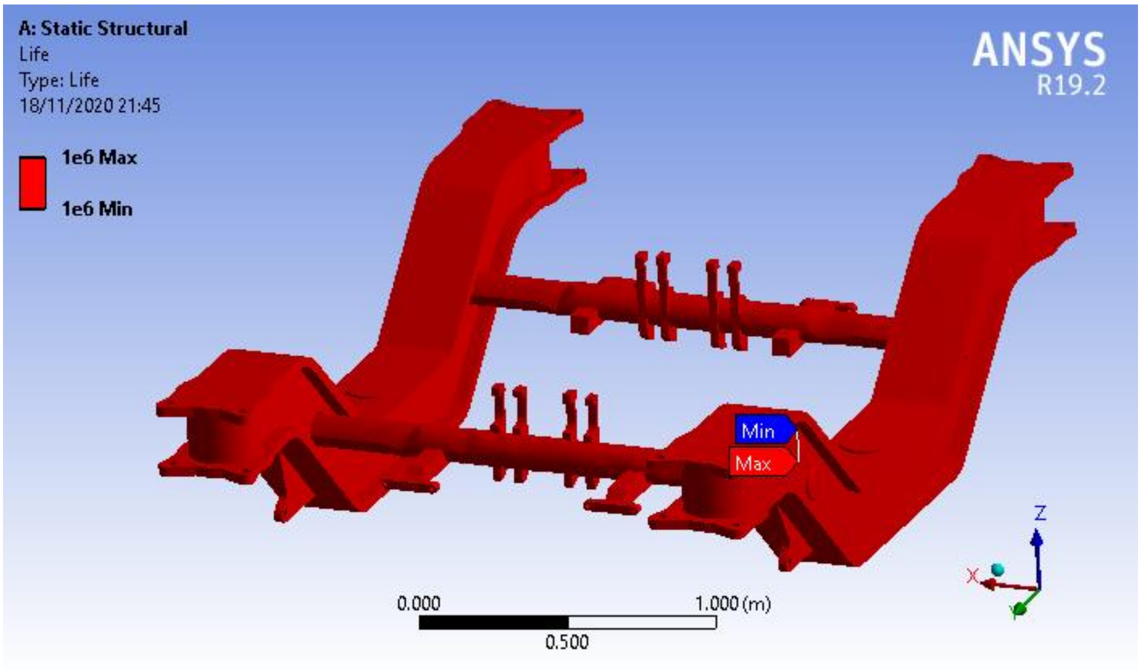


Figure 53. Life expectancy (fatigue) - structure steel - case 8

Composite material frame:

Case 1 - Vertical loads – Boundary conditions – Composite frame;

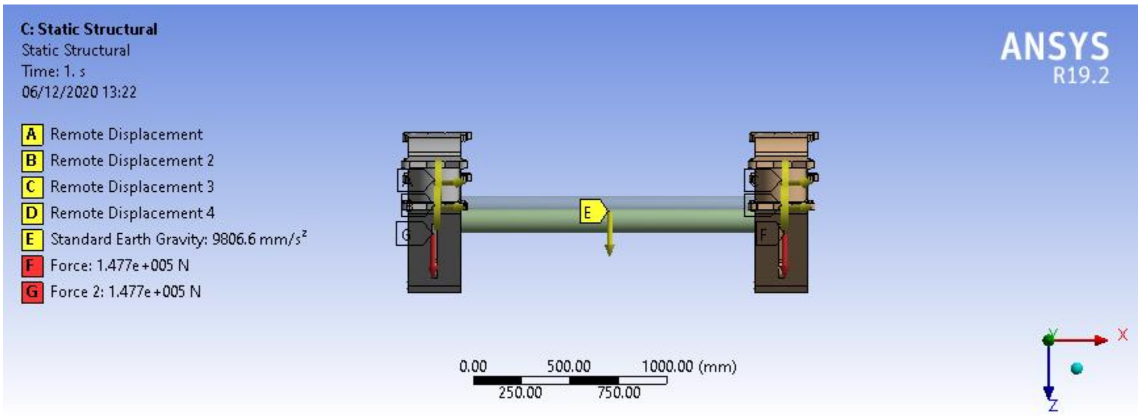


Figure 54. Boundary and forces for CFRP case number 1 - view 1.

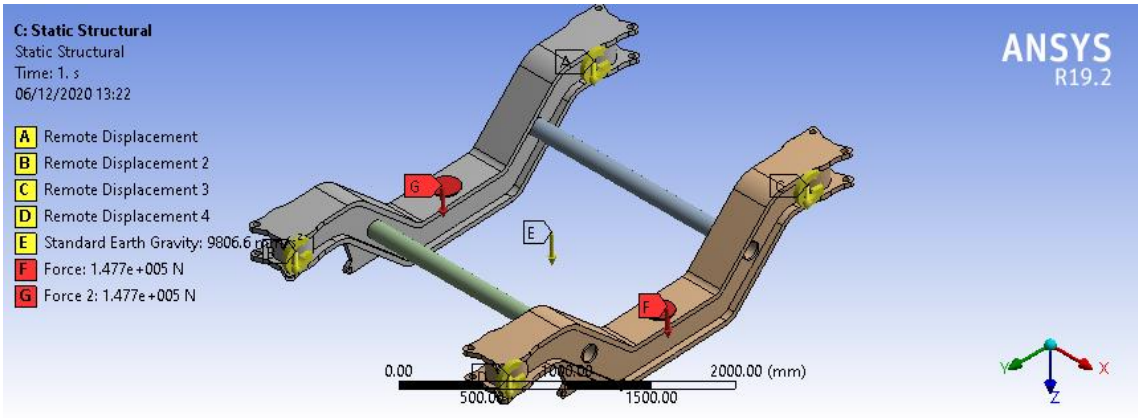


Figure 55. Boundary and forces for CFRP case number 1 - view 2.

Table 9. Detailed table of loads and restrictions for case 1 of the CFRP.

Object Name	*Remote Displacement*	*Remote Displacement 2*	*Remote Displacement 3*	*Remote Displacement 4*	*Force*	*Force 2*
Definition						
Type	Remote Displacement				Force	
X Component	0.01. m (ramped)					
Y Component	0.01. m (ramped)					
Z Component	0. mm (ramped)					
Rotation X	0.1 ° (ramped)					
Rotation Y	0. ° (ramped)					
Rotation Z	0. ° (ramped)					
Suppressed	No					
Behavior	Deformable					
Rotation X		0.1 ° (ramped)				
Rotation Y		0. ° (ramped)				
Rotation Z		0. ° (ramped)				
Rotation X			0.1 ° (ramped)			
Rotation Y			0. ° (ramped)			
Rotation Z			0. ° (ramped)			
Rotation X				0.1 ° (ramped)		
Rotation Y				0. ° (ramped)		
Rotation Z				0. ° (ramped)		
Define By					Vector	
Magnitude					1.477e+005 N (ramped)	
Advanced						
Pinball Region	All					

Case 1 - Vertical loads – Results – Composite frame;

Total deformation:

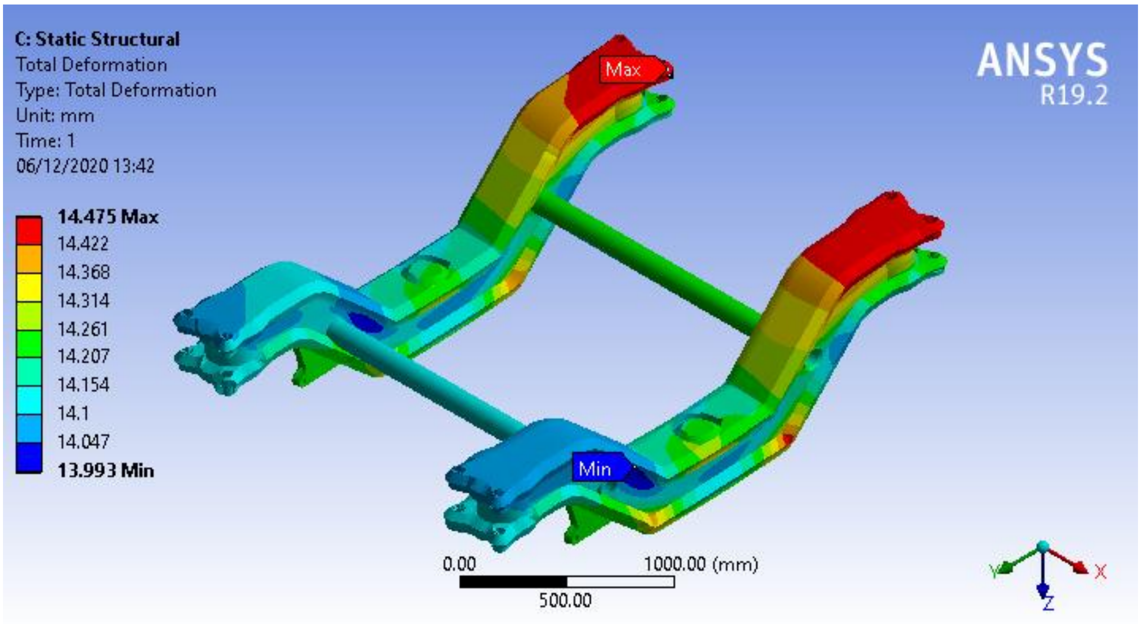


Figure 56. Total deformation 1 - CFRP - case 1.

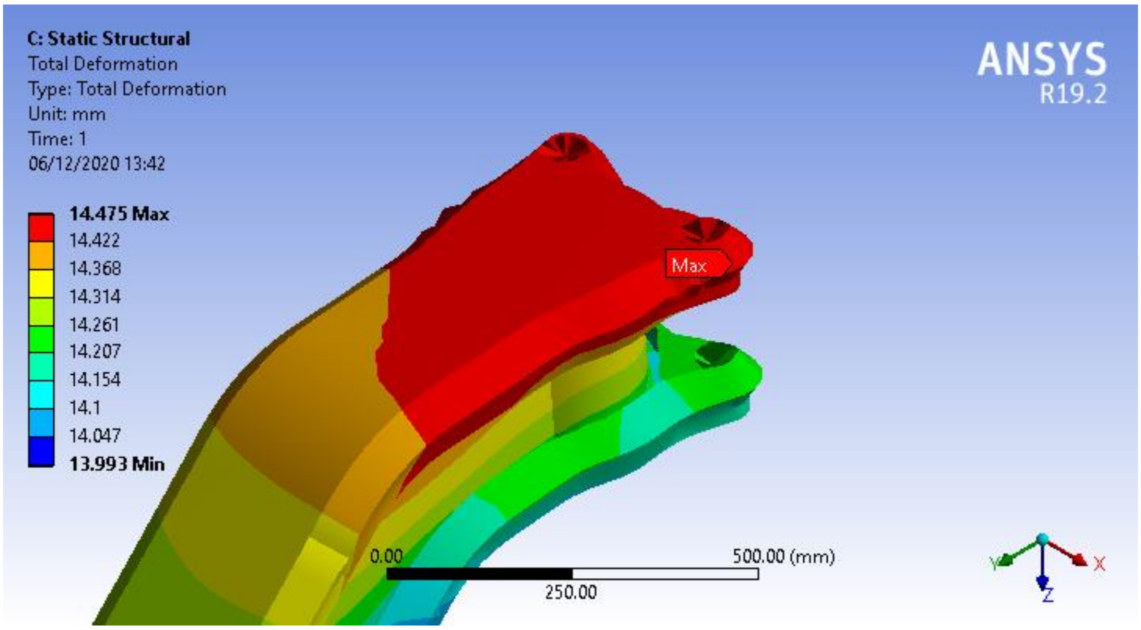


Figure 57. Total deformation 2 - CFRP - case 1

Von-Misses stress:

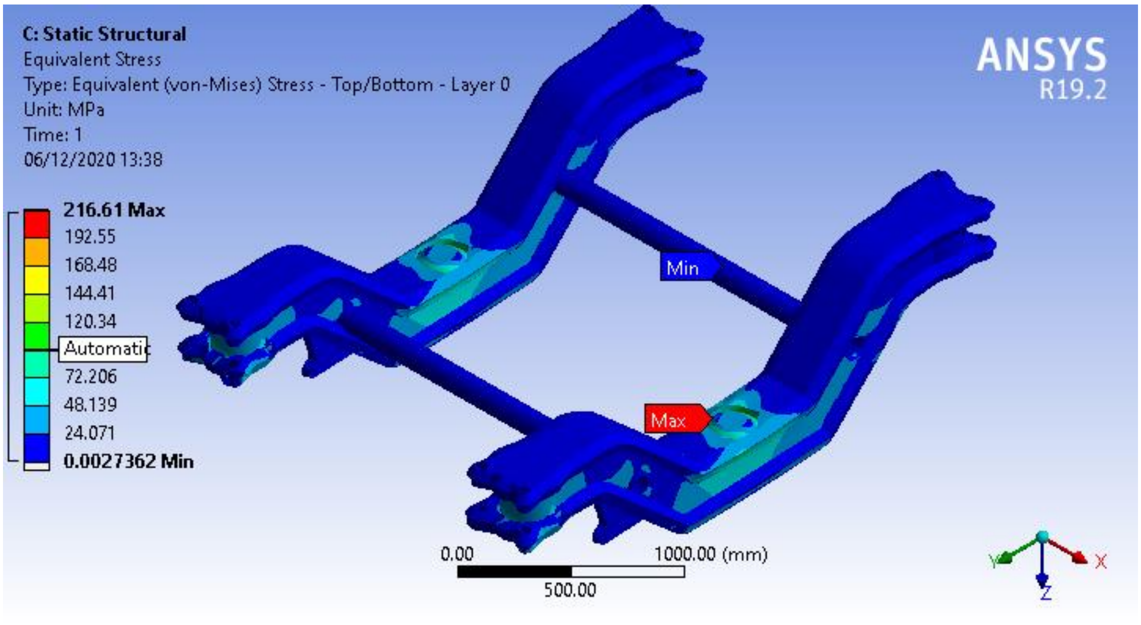


Figure 58. Von-Misses stress 1 - CFRP - case 1

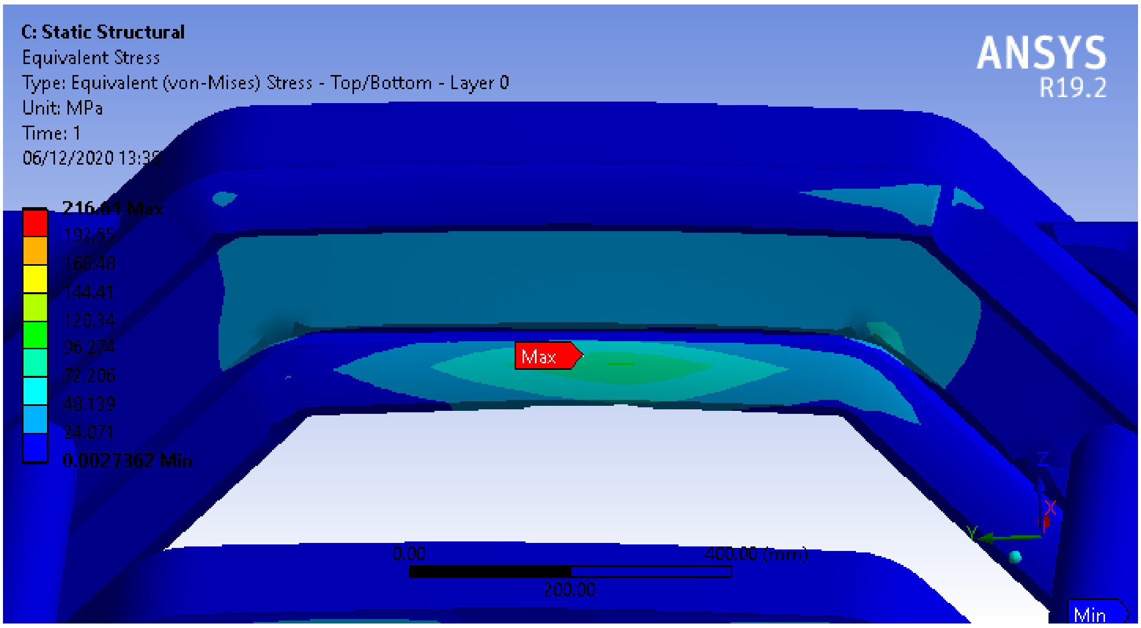


Figure 59. Von-Misses stress 2 - CFRP - case 1

Inverse fail factor:

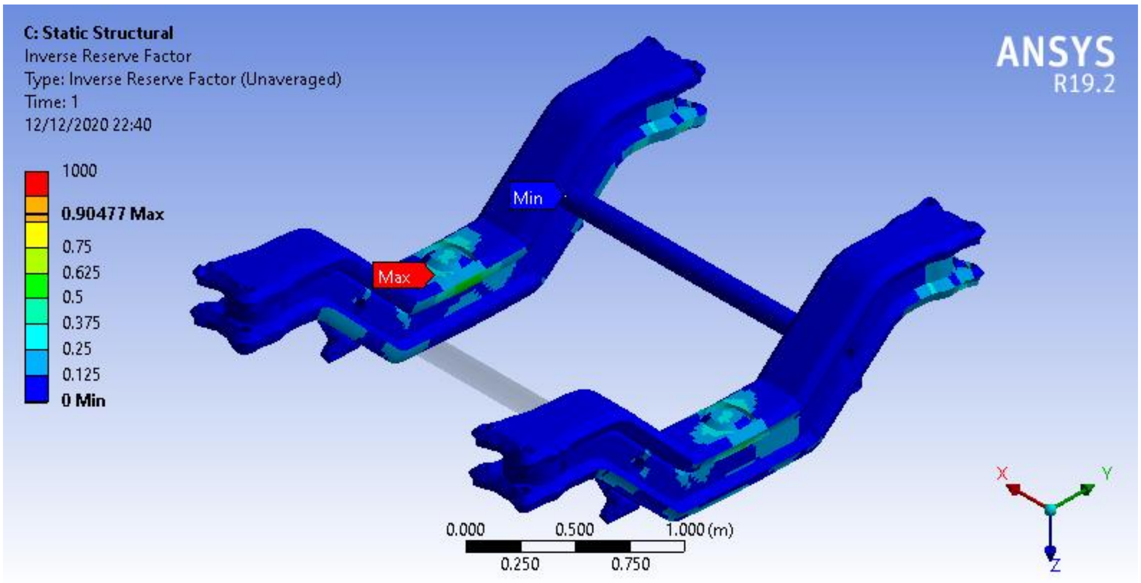


Figure 60. Inverse fail factor 1 - CFRP - case 1

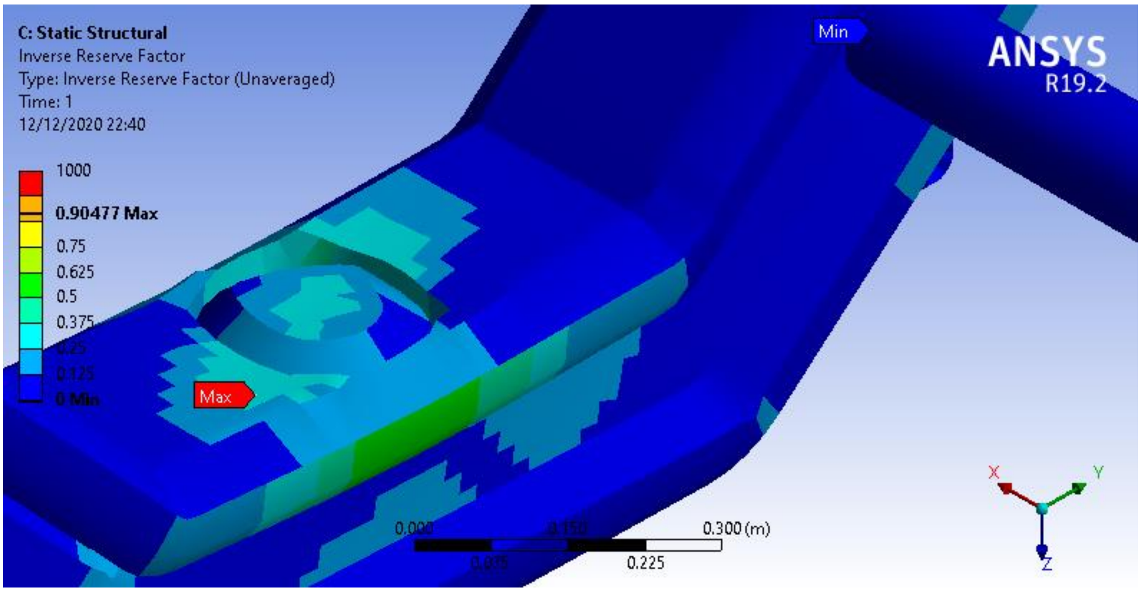


Figure 61. Inverse fail factor 2 - CFRP - case 1

Safety factor:

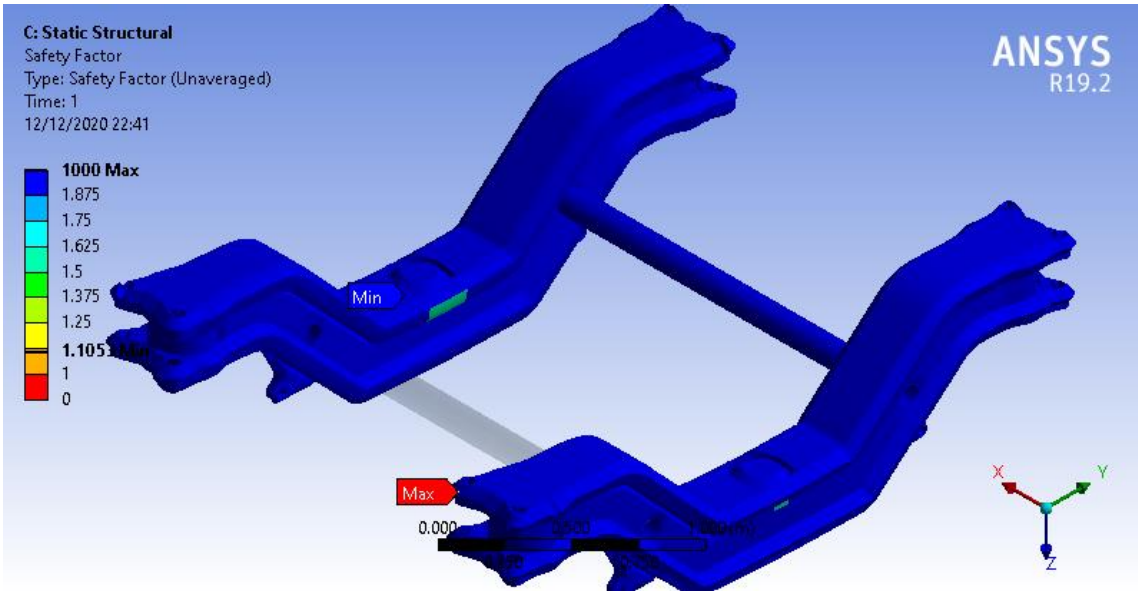


Figure 62. Safety factor - CFRP - case 1

Case 2 - Lateral loads – Boundary conditions – Composite Frame;

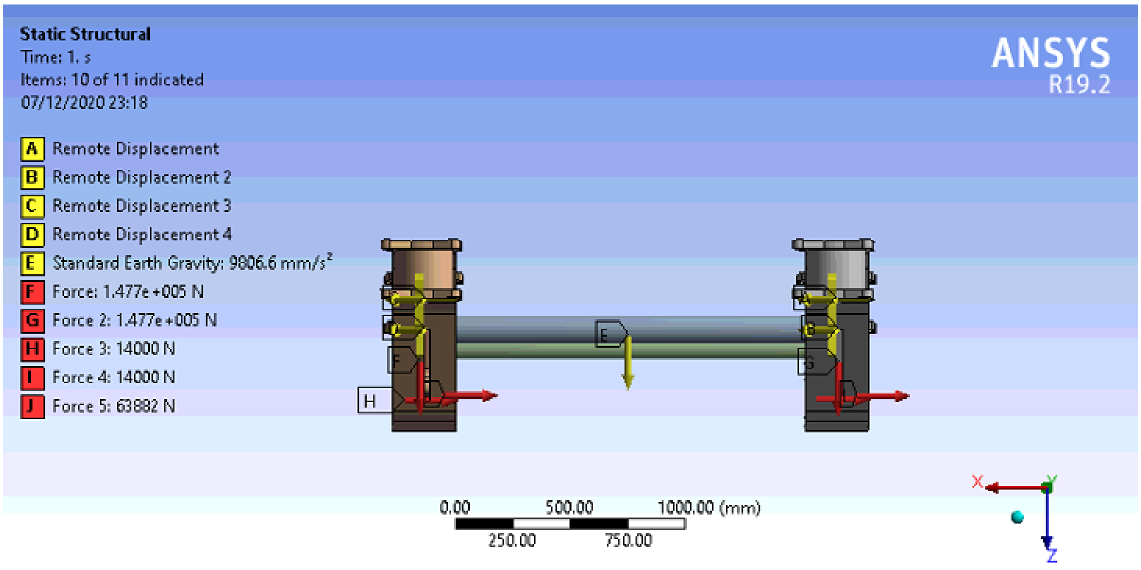


Figure 63. Boundary and forces for CFRP case number 2 - view 1.

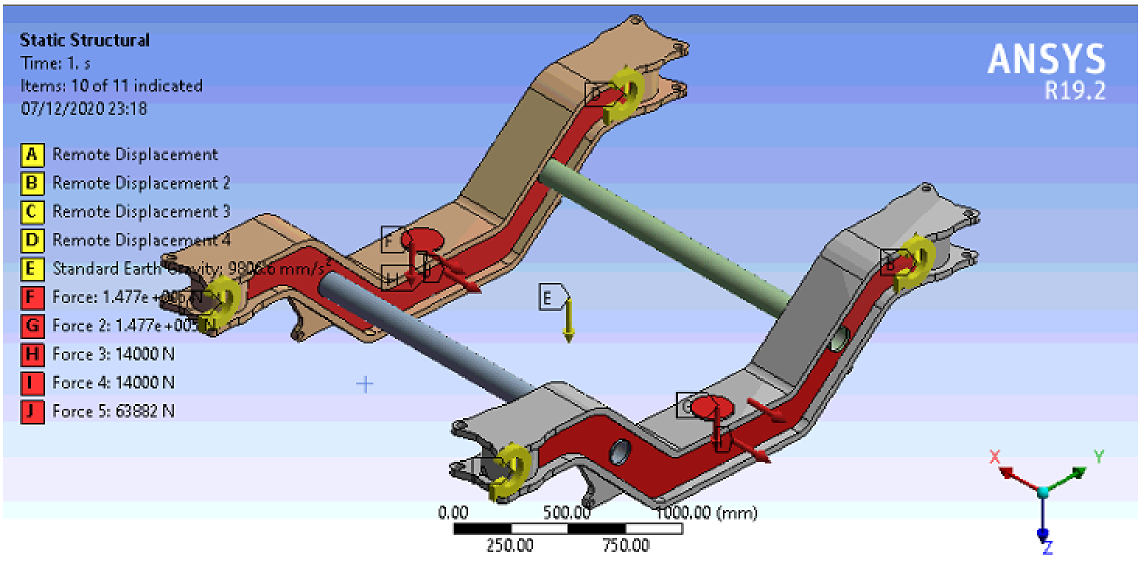


Figure 64. Boundary and forces for CFRP case number 2 - view 2.

Table 10. Detailed table of loads and restrictions for case 2 of the CFRP.

Object Name	Remote Displacement	Remote Displacement 2	Remote Displacement 3	Remote Displacement 4	Force	Force 2	Force 3	Force 4	Force 5	Force 6
Definition										
Type	Remote Displacement				Force					
X Component	0.01. m (ramped)				1.477e+005 N (ramped)		14000 N (ramped)		63882 N (ramped)	
Y Component	0.01. m (ramped)									
Z Component	0. mm (ramped)									
Rotation X	0.1 ° (ramped)									
Rotation Y	0. ° (ramped)									
Rotation Z	0. ° (ramped)									
Suppressed	No									
Behavior	Deformable									
Rotation X		0.1 ° (ramped)								
Rotation Y		0. ° (ramped)								
Rotation Z		0. ° (ramped)								
Rotation X			0.1 ° (ramped)							
Rotation Y			0. ° (ramped)							
Rotation Z			0. ° (ramped)							
Rotation X				0.1 ° (ramped)						
Rotation Y				0. ° (ramped)						
Rotation Z				0. ° (ramped)						
Define By										
Magnitude										
Advanced										
Pinball Region	All									

Case 2 - Lateral loads – Results – Composite Frame;

Total deformation:

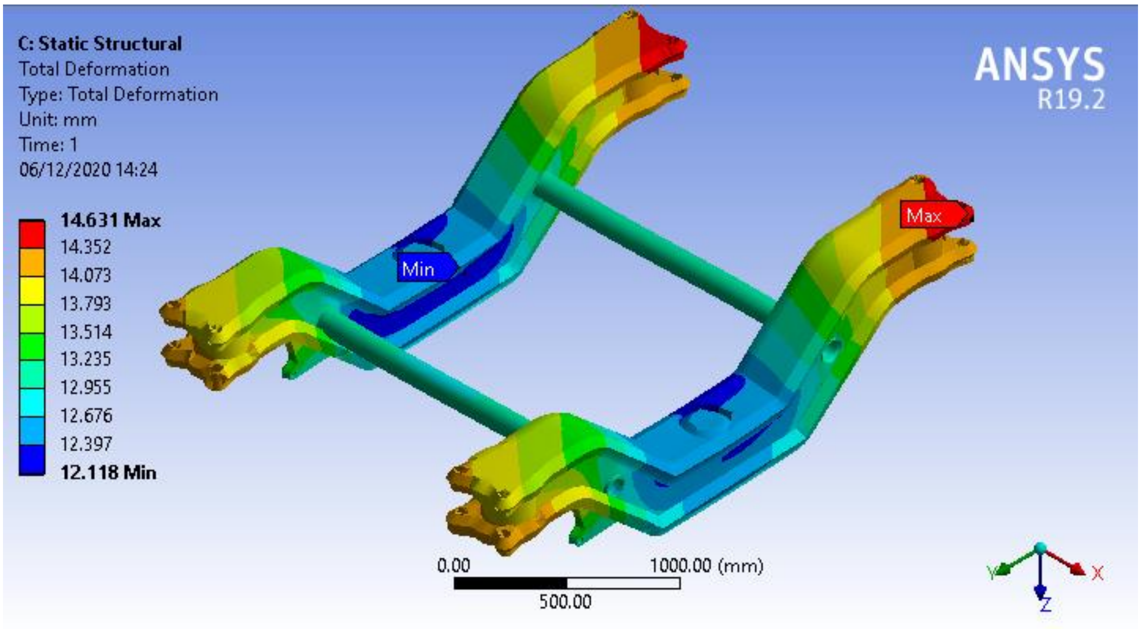


Figure 65. Total deformation 1 - CFRP - case 2.

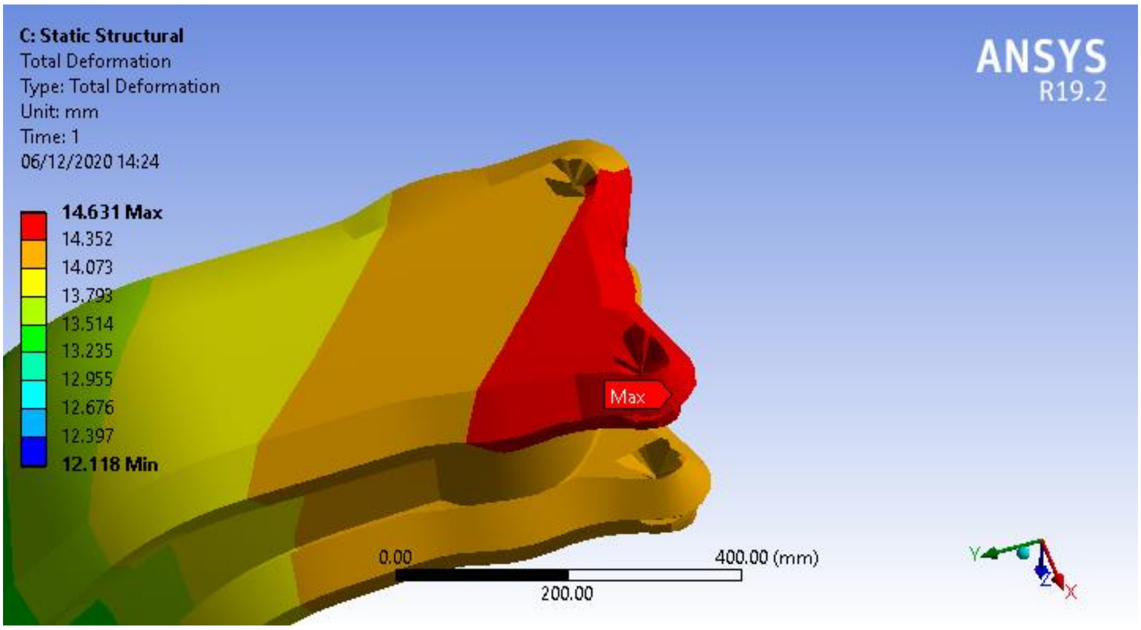


Figure 66. Total deformation 2 - CFRP - case 2.

Von-Misses stress:

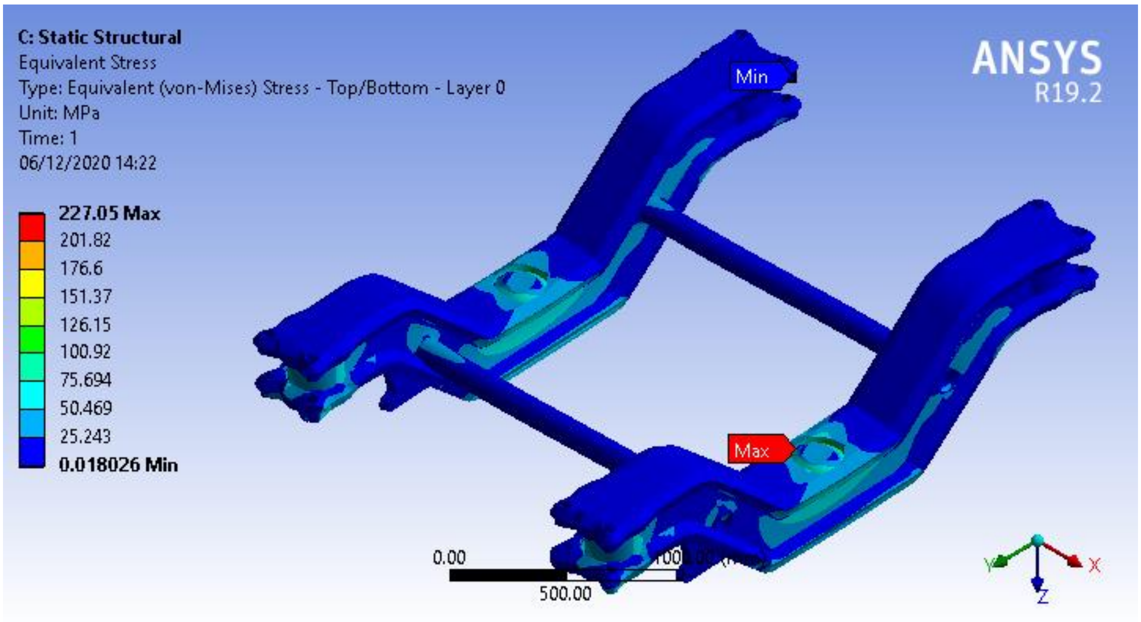


Figure 67. Von-misses stress 1 - CFRP - case 2.

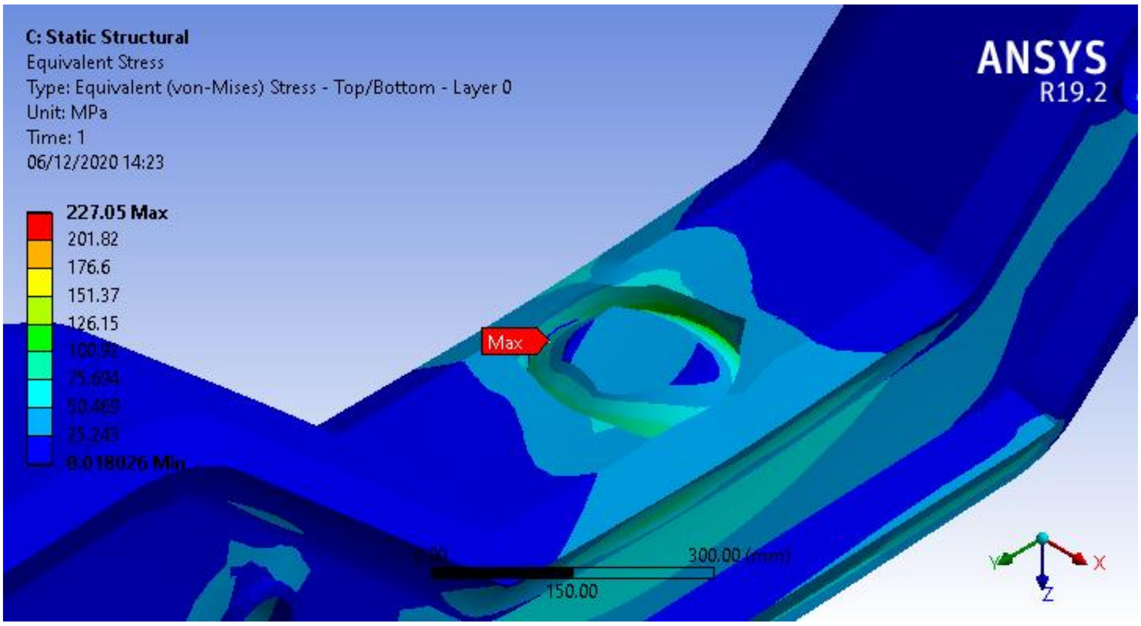


Figure 68. Von-misses stress 2 - CFRP - case 2.

Inverse fail factor:

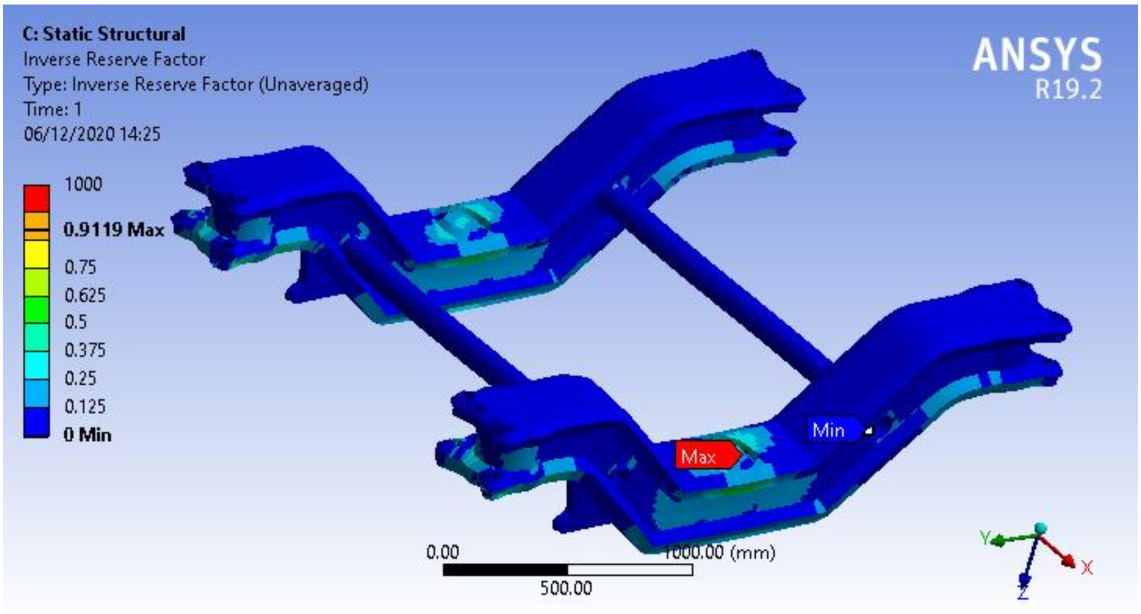


Figure 69. Inverse reverse fails 1 - CFRP - case 2.

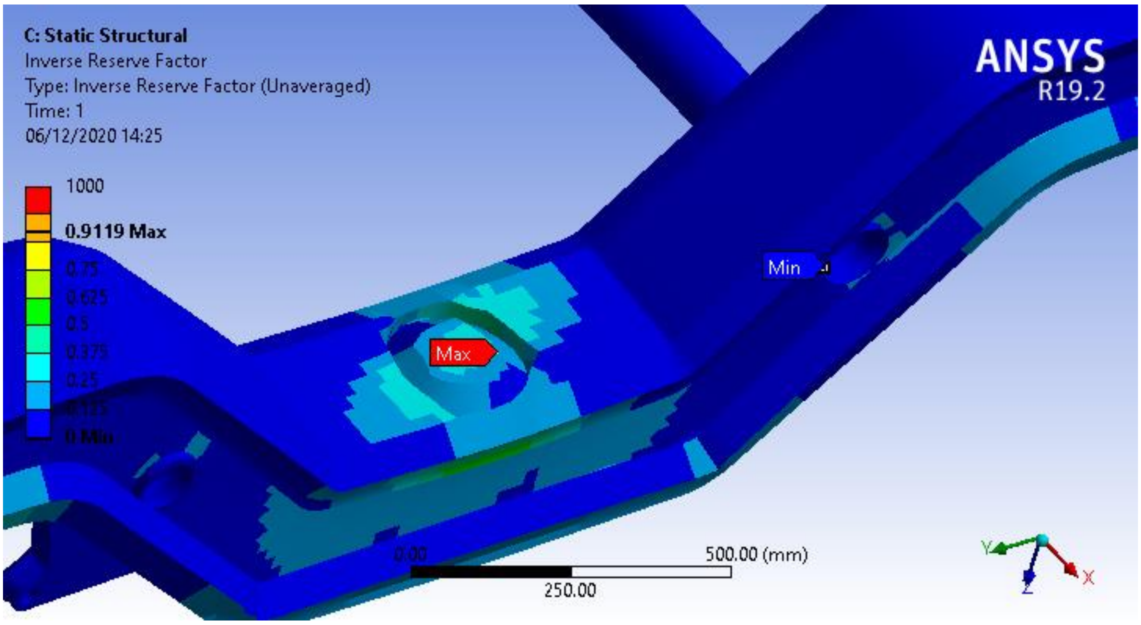


Figure 70. Inverse reverse fails 2 - CFRP - case 2.

Safety Factor:

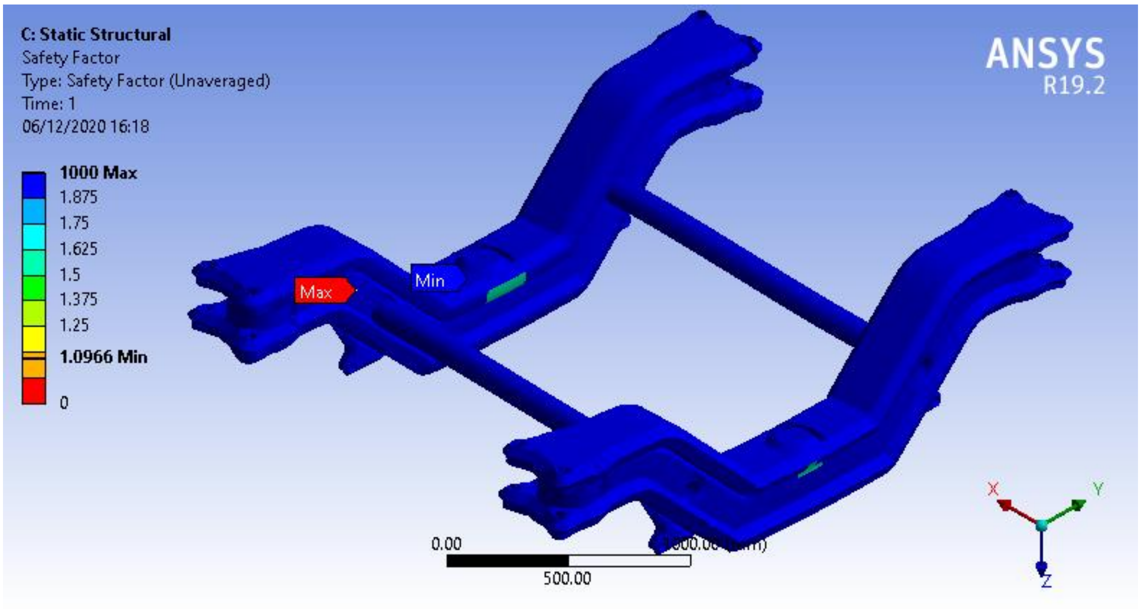


Figure 71. Safety factor - CFRP - case 2.

Case 3 - Unloading loads – Boundary conditions – Composite Frame;

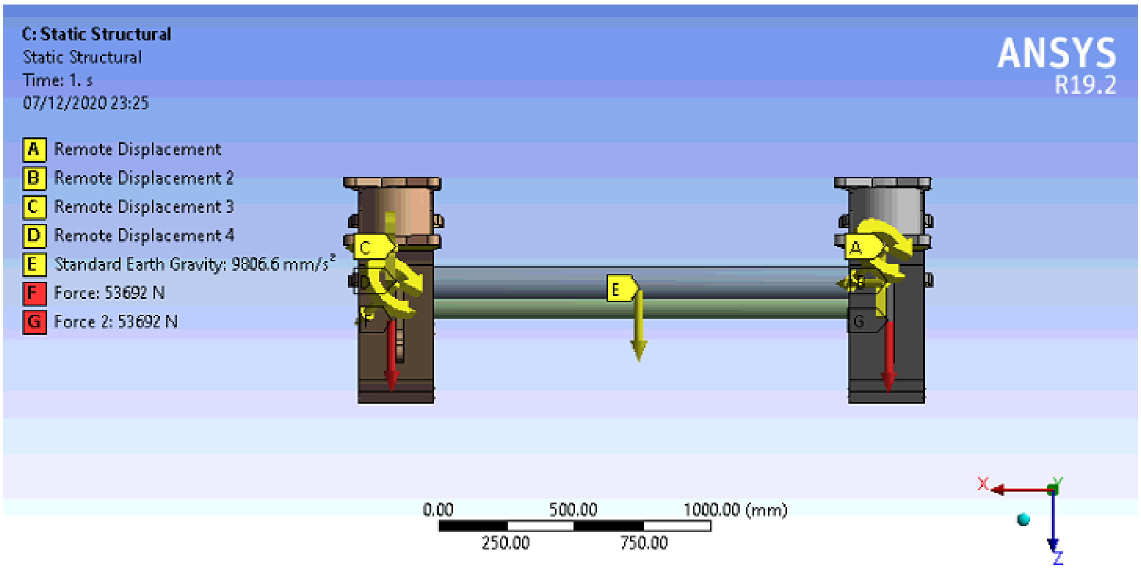


Figure 72. Boundary and forces for CFRP case number 3 - view 1.

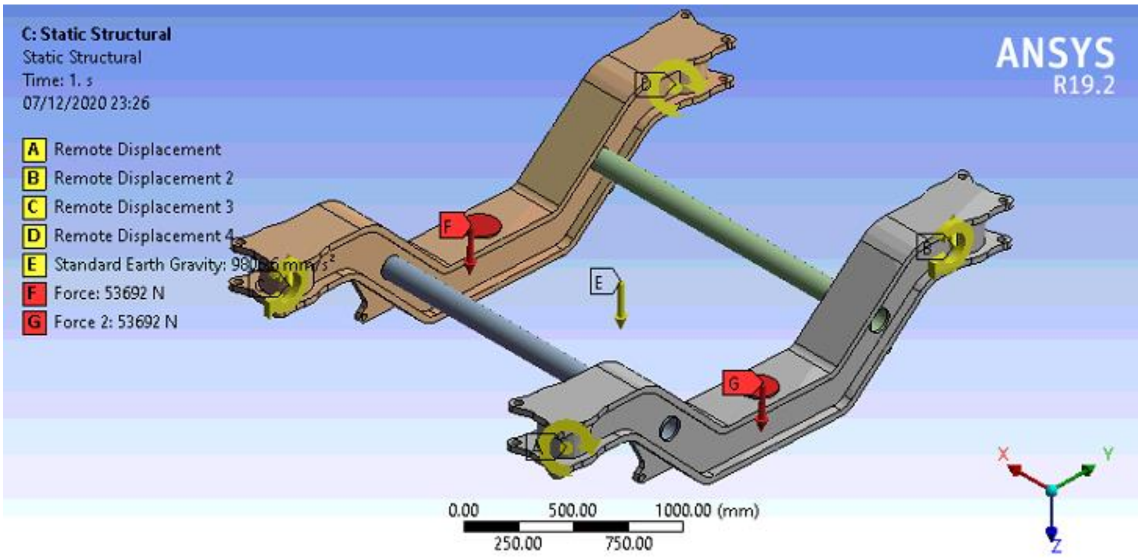


Figure 73.Boundary and forces for CFRP case number 3 - view 2.

Table 11. Detailed table of loads and restrictions for case 3 of the CFRP.

Object Name	*Remote Displacement*	*Remote Displacement 2*	*Remote Displacement 3*	*Remote Displacement 4*	*Force*	*Force 2*
State	Fully Defined					
Definition						
Type	Remote Displacement				Force	
X Component	0.01. m (ramped)					
Y Component	0.01. m (ramped)					
Z Component	10. mm (ramped)	0. mm (ramped)		10. mm (ramped)		
Rotation X	0.1 ° (ramped)					
Rotation Y	0.1 ° (ramped)					
Rotation Z	0.1 ° (ramped)					
Suppressed	No					
Behavior	Deformable					
Rotation X		0.1 ° (ramped)				
Rotation Y		0. ° (ramped)				
Rotation Z		0. ° (ramped)				
Rotation X			0.1 ° (ramped)			
Rotation Y			0. ° (ramped)			
Rotation Z			0. ° (ramped)			
Rotation X				0.1 ° (ramped)		
Rotation Y				0.1 ° (ramped)		
Rotation Z				0.1 ° (ramped)		
Define By					Vector	
Magnitude					53692 N (ramped)	
Advanced						
Pinball Region	All					

Case 3 - Unloading loads – Results – Composite Frame;

Total deformation:

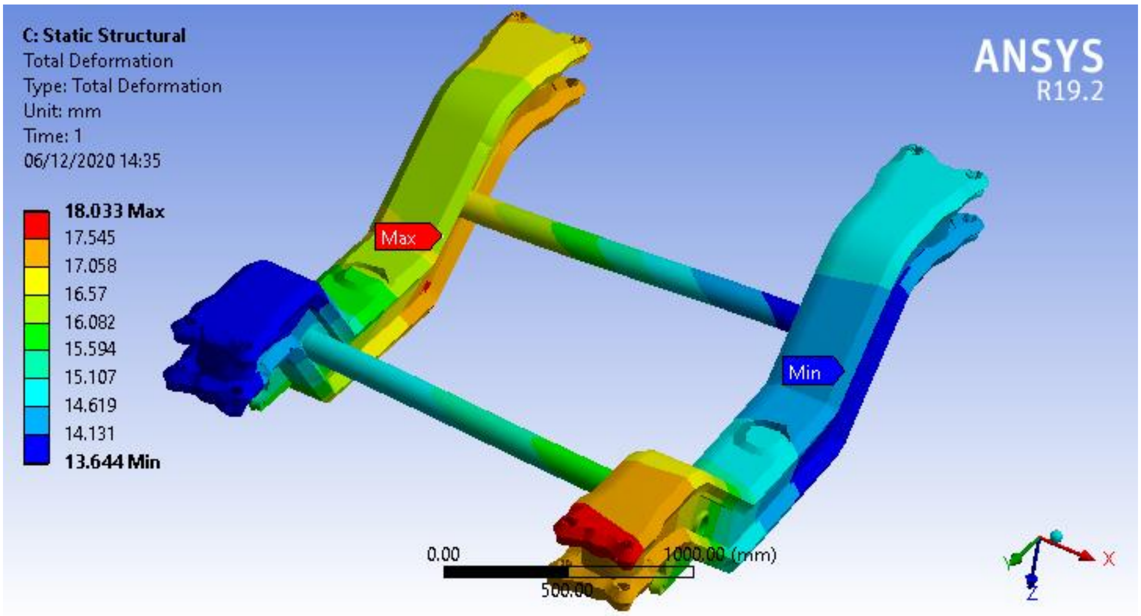


Figure 74. Total deformation 1 - CFRP - case 3.

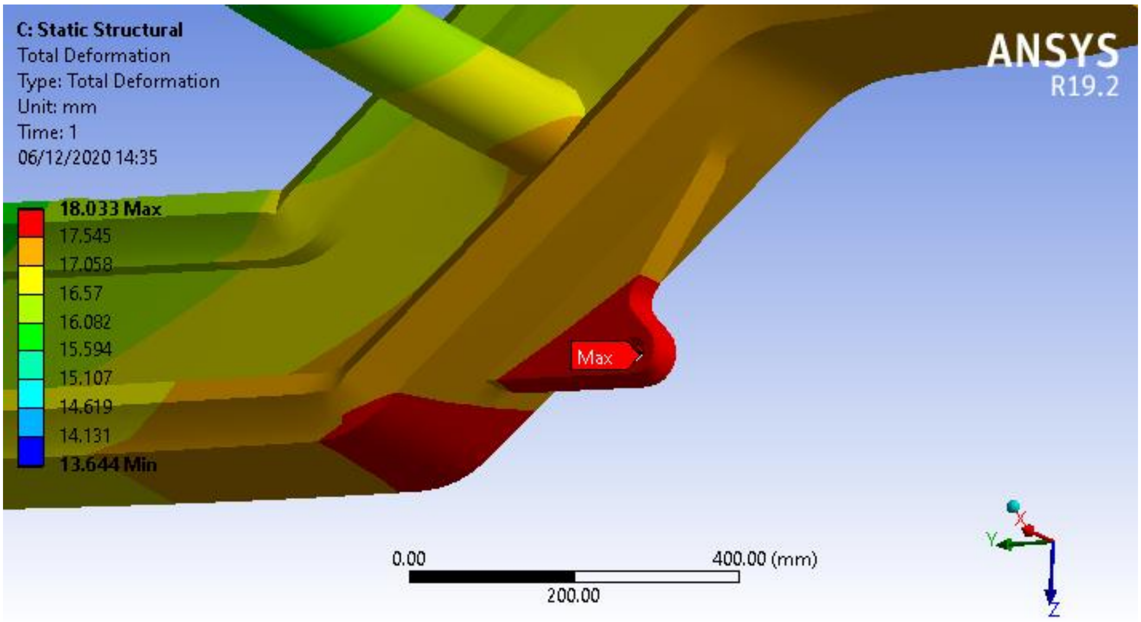


Figure 75. Total deformation 2 - CFRP - case 3.

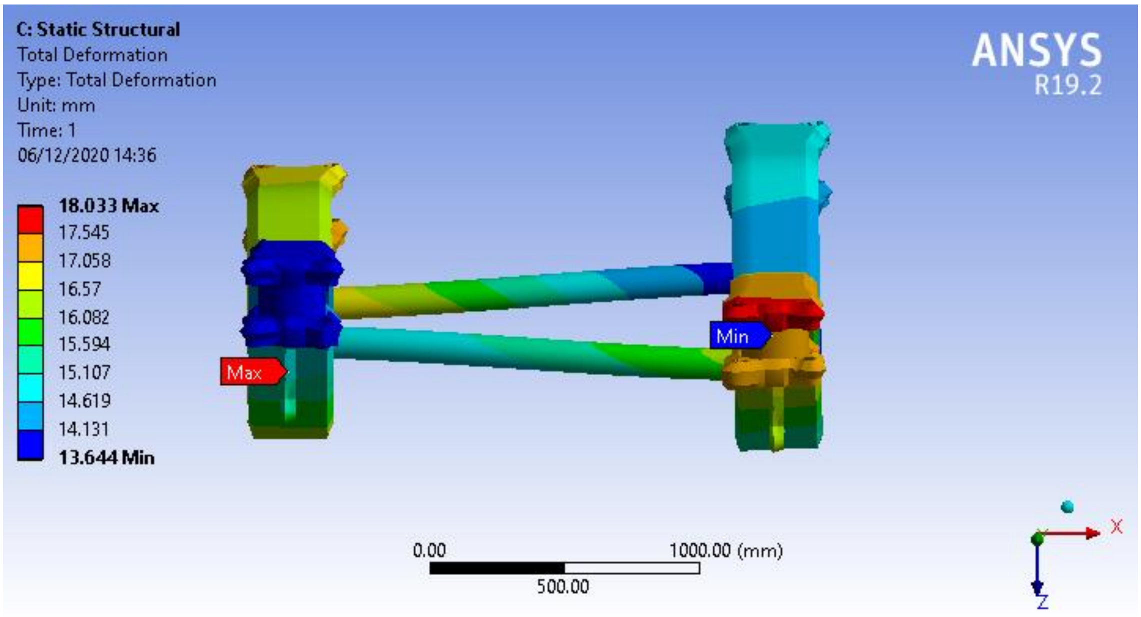


Figure 76. Total deformation 3 - CFRP - case 3.

Von-Misses stress:

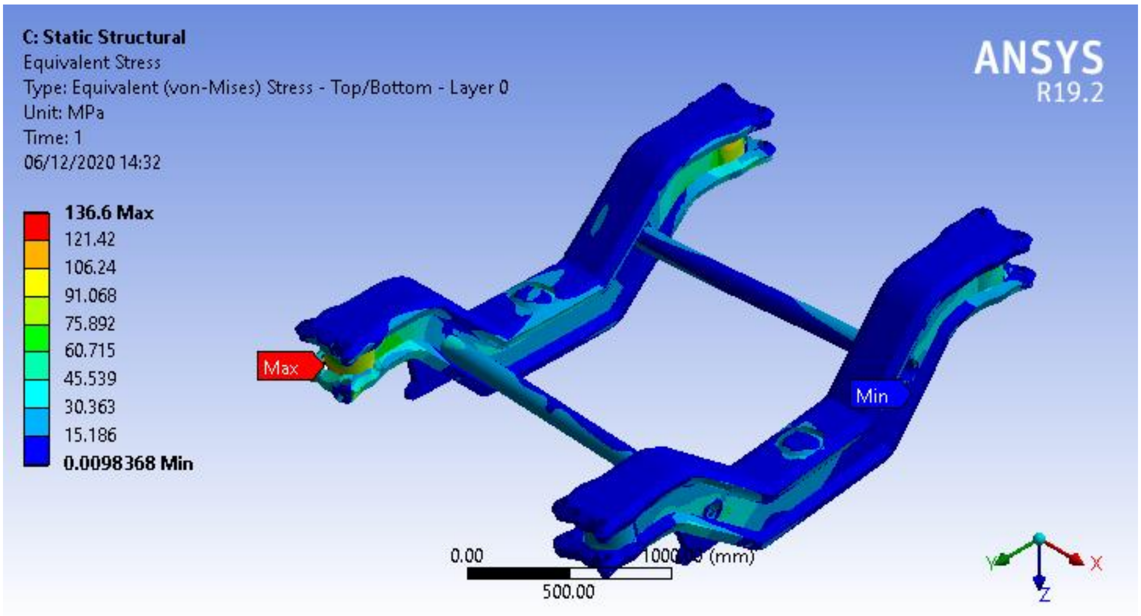


Figure 77. Von-Misses stress 1 – CFRP - case 3.

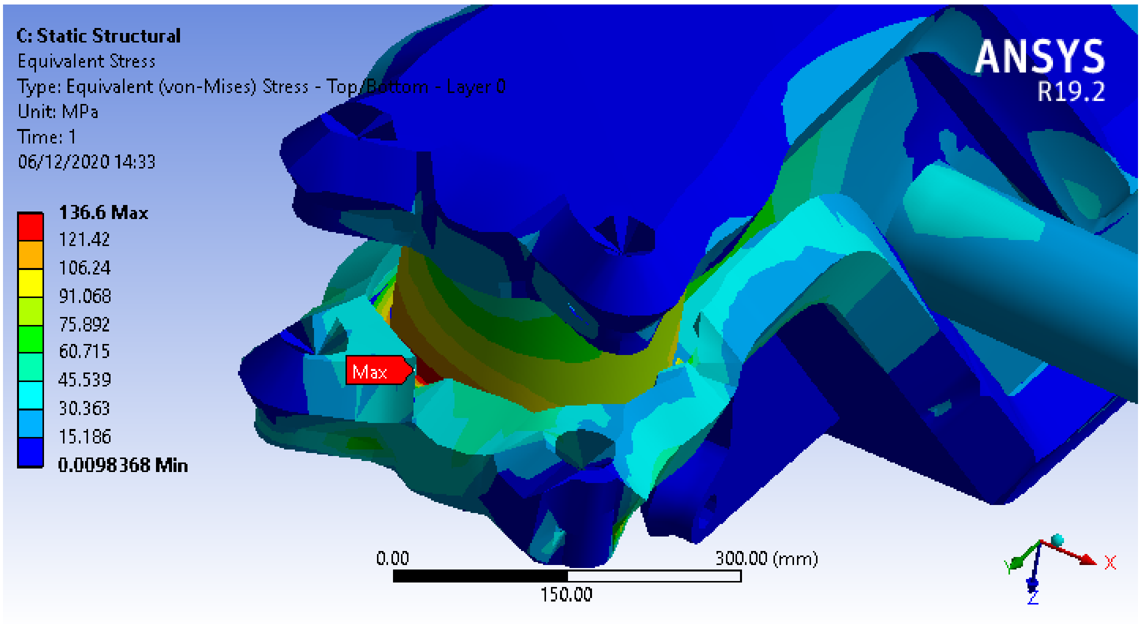


Figure 78. Von-Misses stress 2 - CFRP - case 3.

Inverse fail factor:

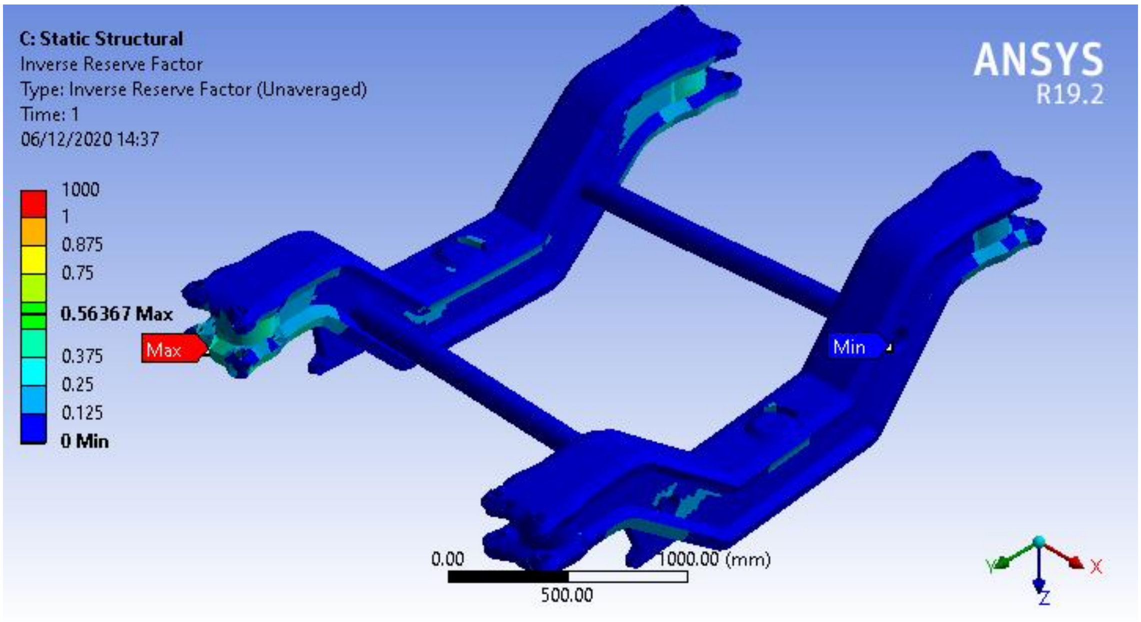


Figure 79. Inverse fail factor 1 - CFRP - case 3.

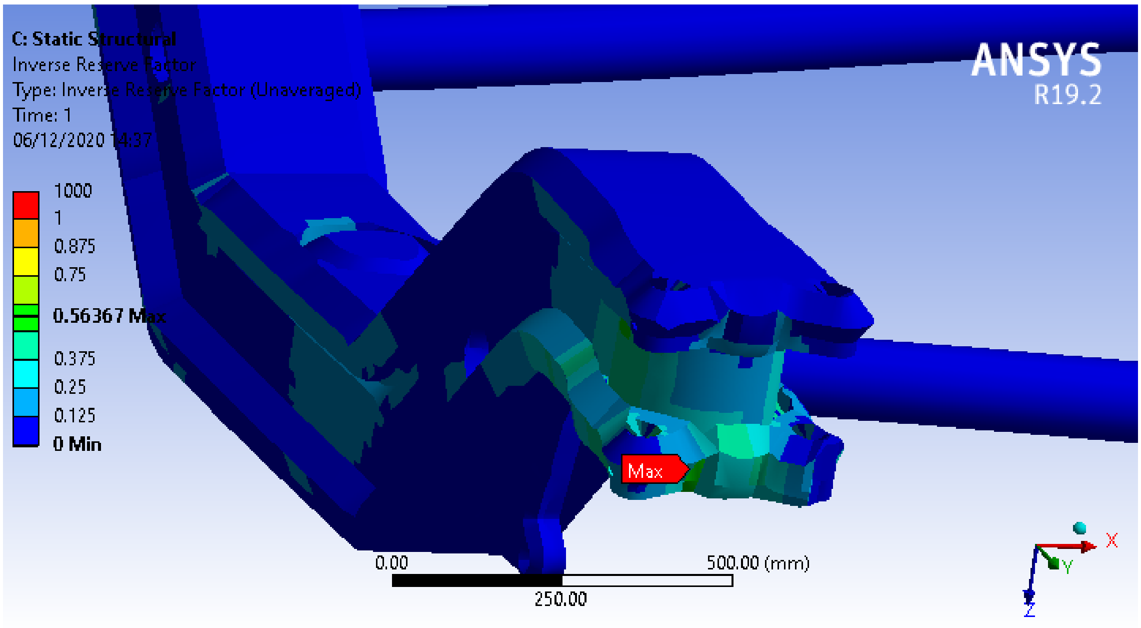


Figure 80. Inverse fail factor 2 - CFRP - case 3.

Safety factor:

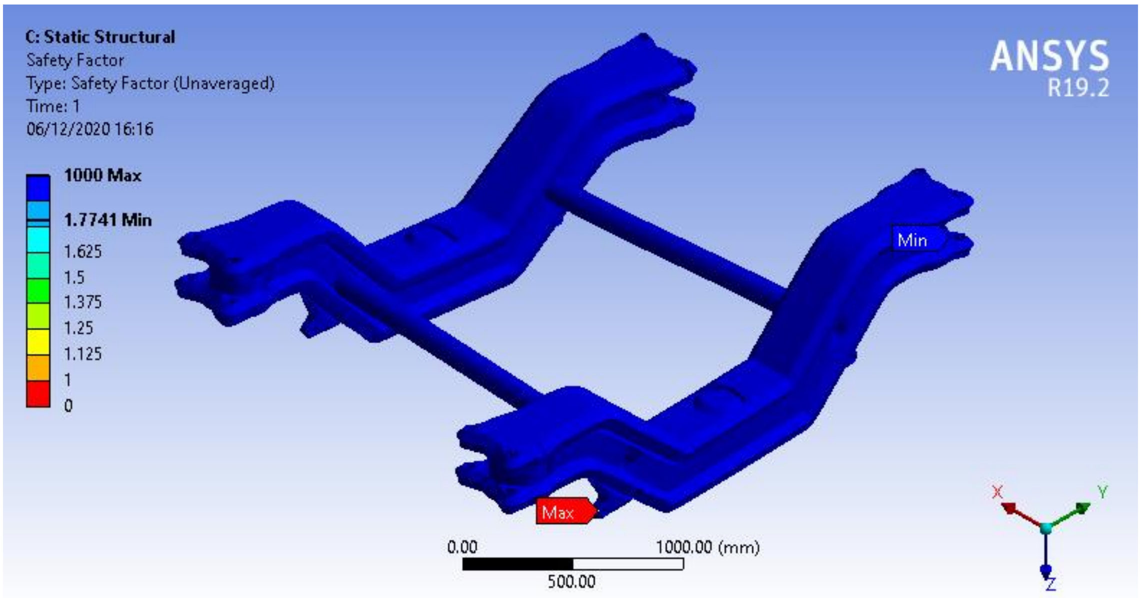


Figure 81. Safety factor - CFRP - case 3.

Case 4 - Lonzening loads – Boundary conditions – Composite Frame;

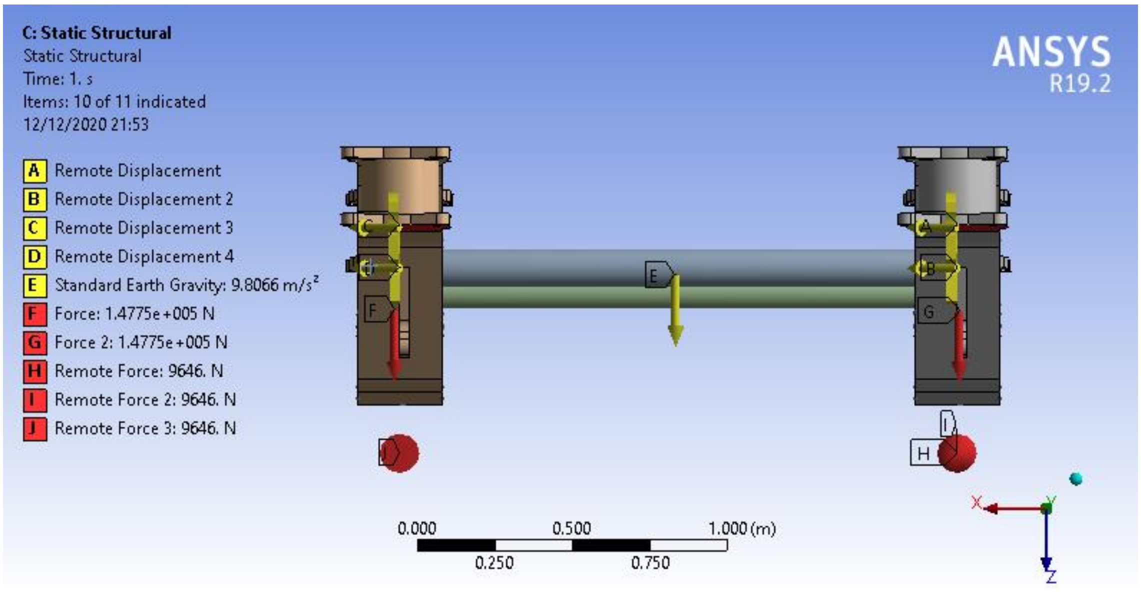


Figure 82. Boundary and forces for CFRP case number 4 - view 1.

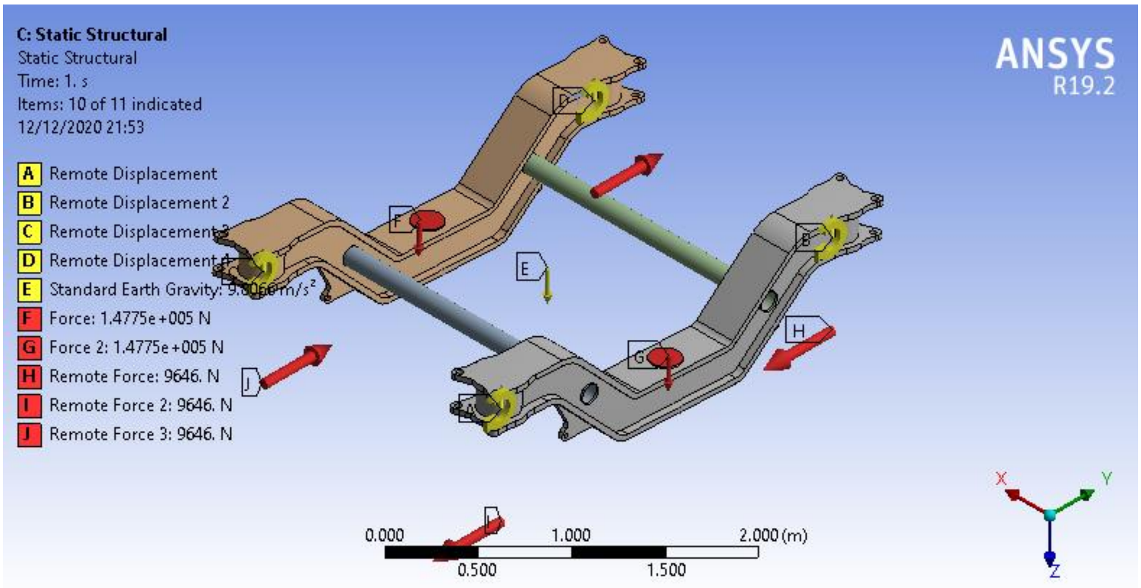


Figure 83. Boundary and forces for CFRP case number 4 - view 2.

Table 12. Detailed table of loads and restrictions for case 4 of the CFRP.

Object Name	*Remote Displacement*	*Remote Displacement 2*	*Remote Displacement 3*	*Remote Displacement 4*	*Force*	*Force 2*	*Force 3*	*Force 4*	*Force 5*	*Force 6*
Definition										
Type	Remote Displacement				Force					
X Component	0.01. m (ramped)									
Y Component	0.01. m (ramped)						-9646. N (ramped)		9646. N (ramped)	
Z Component	0. mm (ramped)				1.4775e+005 N (ramped)					
Rotation X	0.1 ° (ramped)									
Rotation Y	0. ° (ramped)									
Rotation Z	0. ° (ramped)									
Suppressed	No									
Behavior	Deformable									
Rotation X		0.1 ° (ramped)								
Rotation Y		0. ° (ramped)								
Rotation Z		0. ° (ramped)								
Rotation X			0.1 ° (ramped)							
Rotation Y			0. ° (ramped)							
Rotation Z			0. ° (ramped)							
Rotation X				0.1 ° (ramped)						
Rotation Y				0. ° (ramped)						
Rotation Z				0. ° (ramped)						
Define By										
Magnitude										
Advanced										
Pinball Region	All									

Case 4 - Lonzening loads – Results – Composite Frame;

Total deformation:

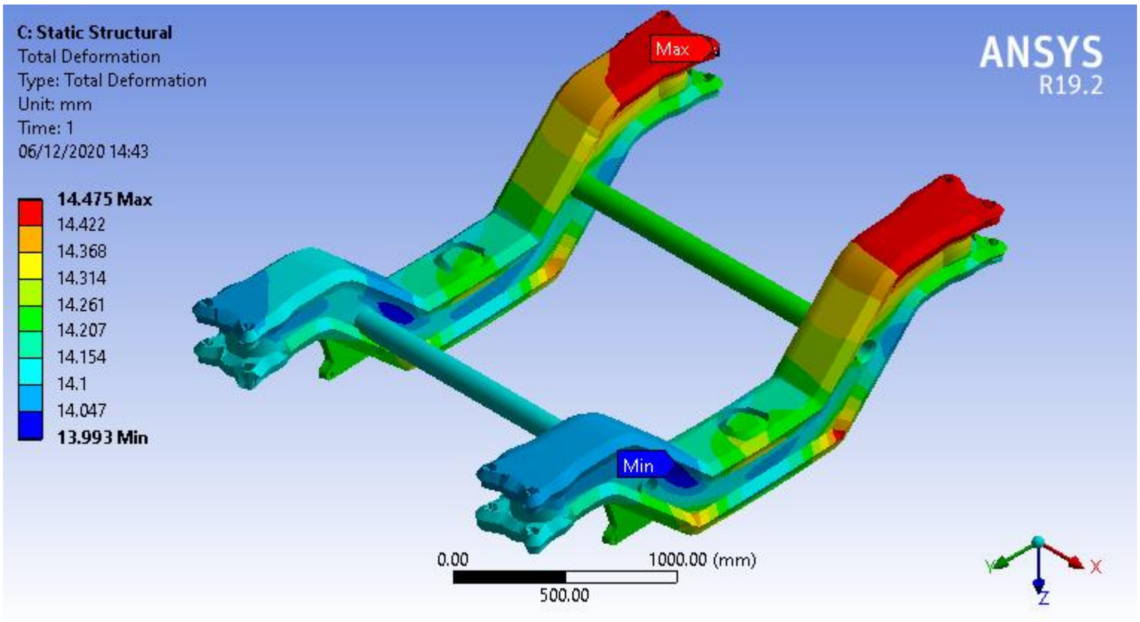


Figure 84. Total deformation 1 - CFRP - case 4.

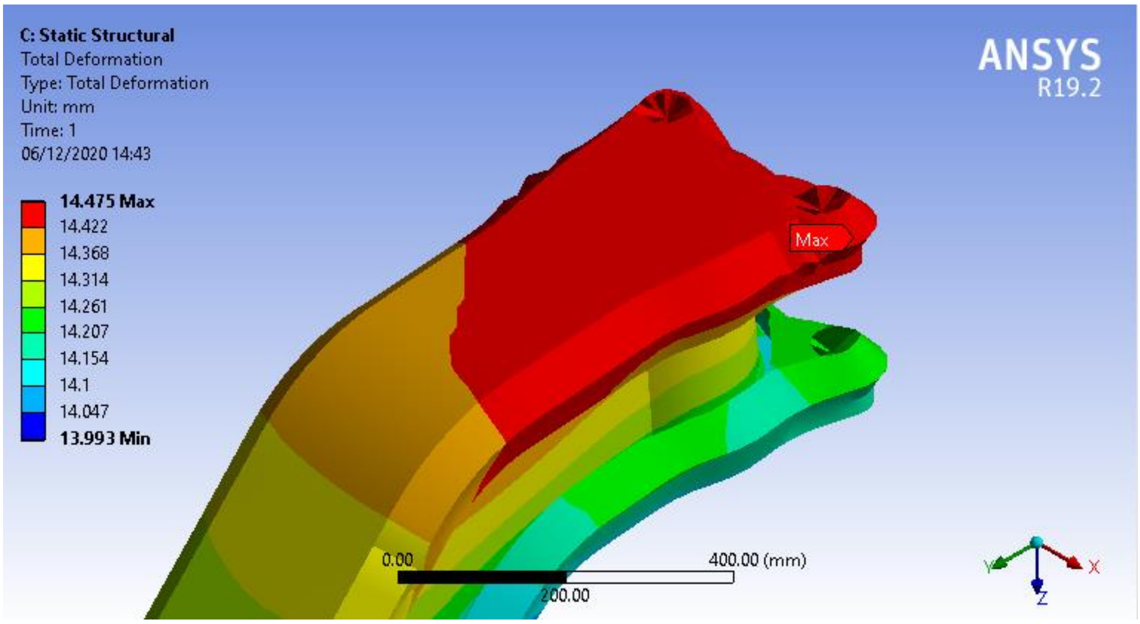


Figure 85. Total deformation 2- CFRP - case 4.

Von-Misses stress:

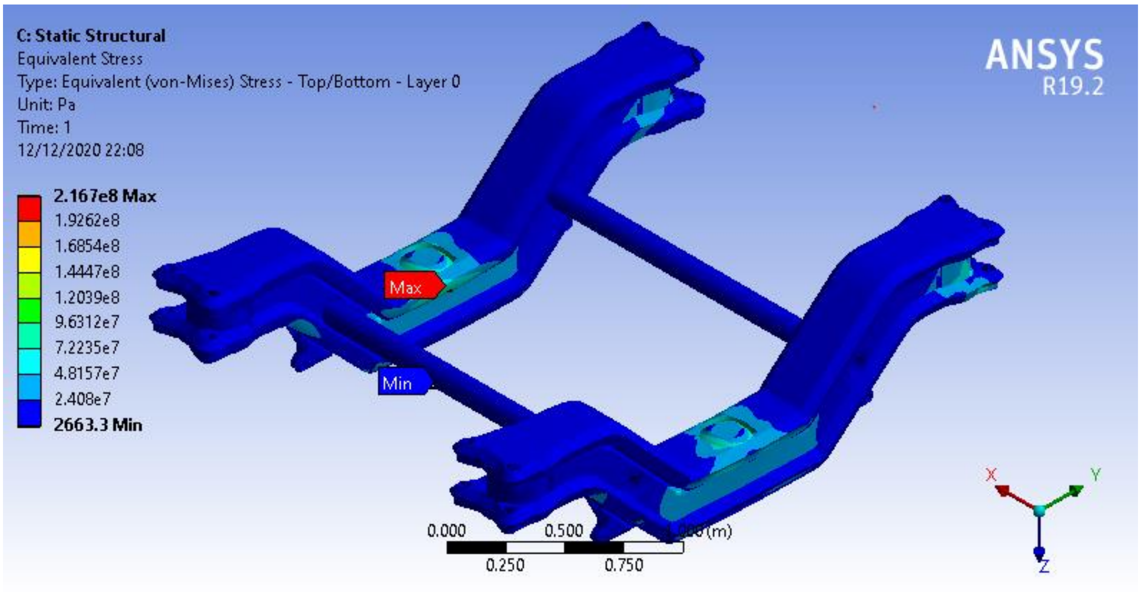


Figure 86. Von-misses stress 1 - CFRP - case 4.

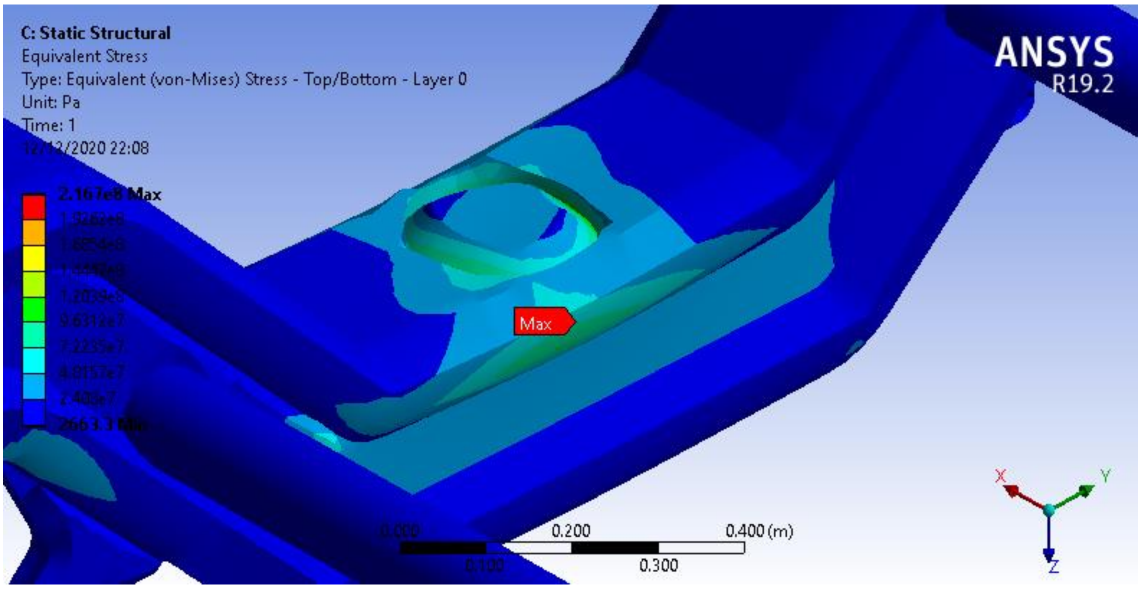


Figure 87. Von-misses stress 2 - CFRP - case 4.

Inverse fail factor:

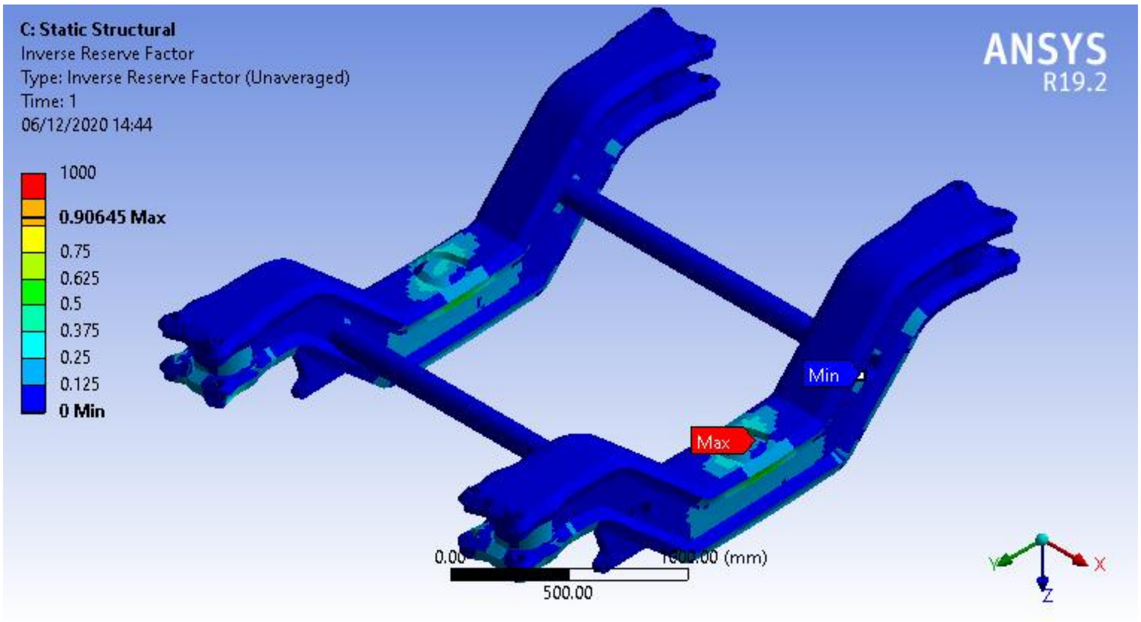


Figure 88. Inverse fail factor 1 - CFRP - case 4.

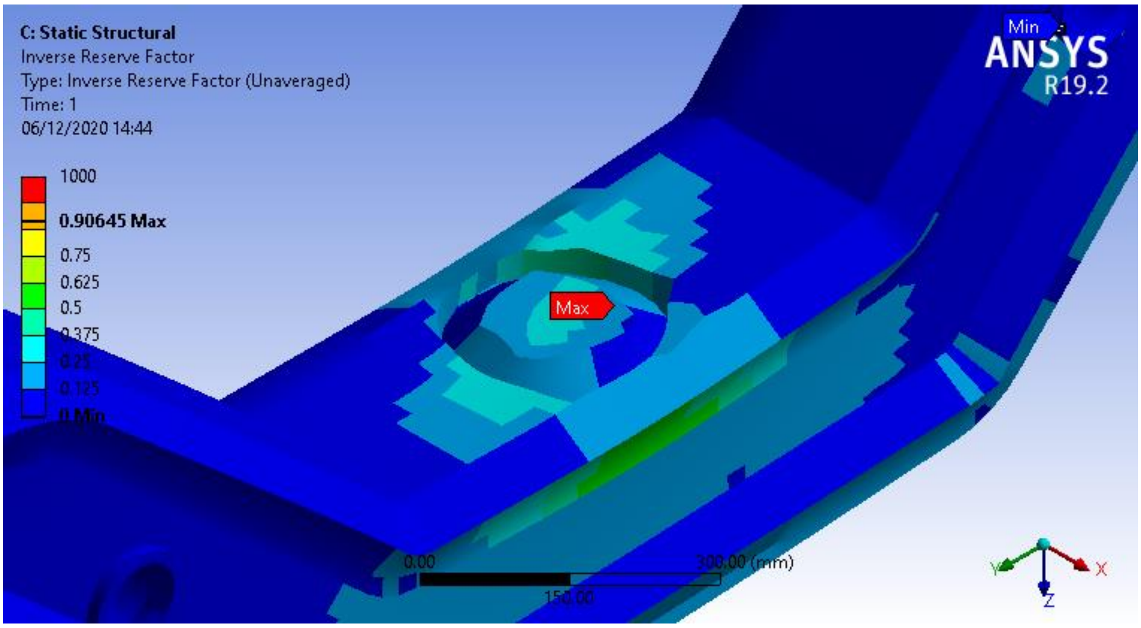


Figure 89. Inverse fail factor 2 - CFRP - case 4.

Safety Factor:

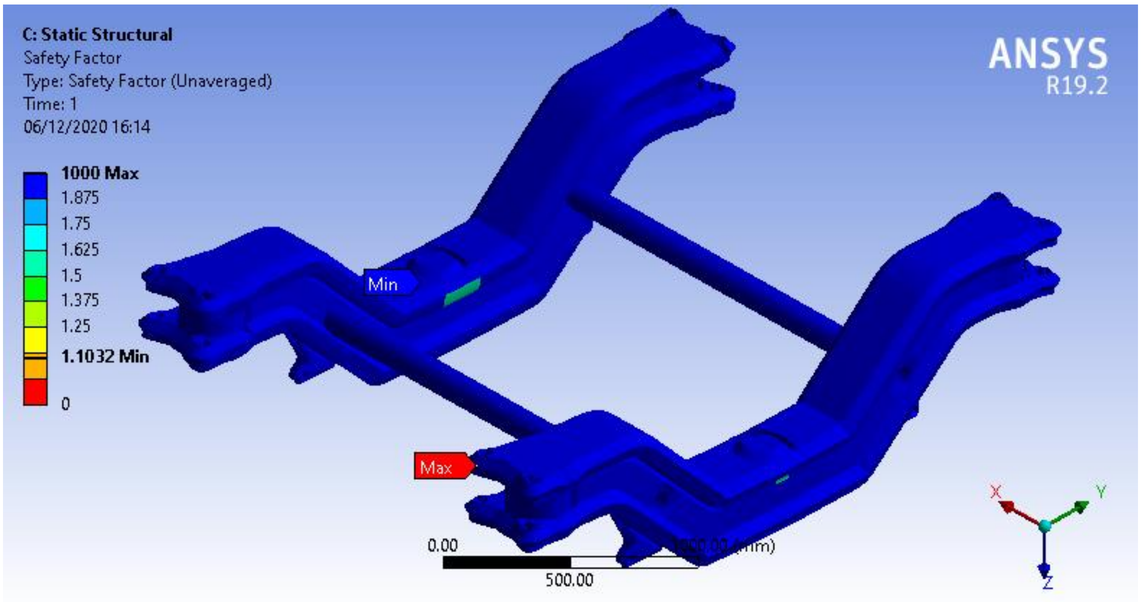


Figure 90. Safety factor - CFRP - case 4.

Case 5 - Longitudinal loads – Boundary conditions – Composite Frame;

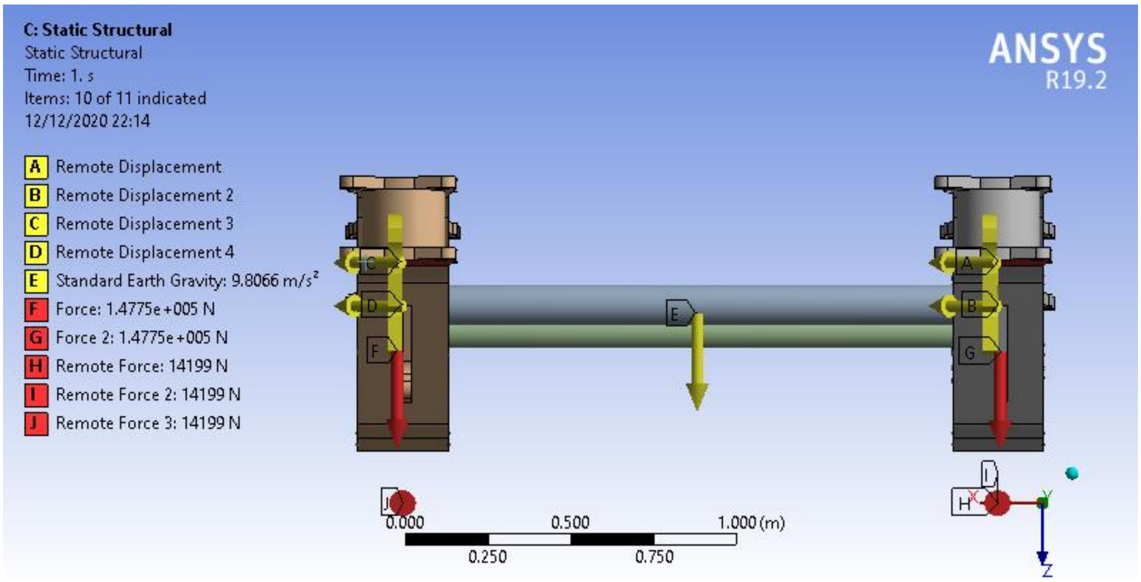


Figure 91. Boundary and forces for CFRP case number 5 - view 1.

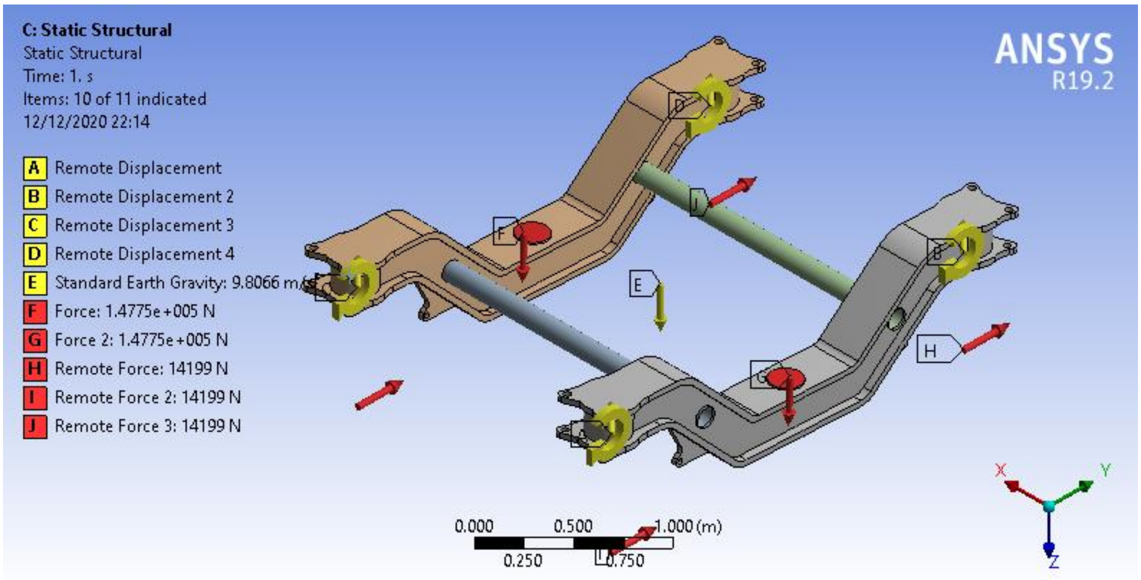


Figure 92.Boundary and forces for CFRP case number 5 - view 2.

Table 13. Detailed table of loads and restrictions for case 5 of the CFRP.

Object Name	*Remote Disp*	*Remote Disp 2*	*Remote Disp 3*	*Remote Disp 4*	*Force*	*Force 2*	*Force 3*	*Force 4*	*Force 5*	*Force 6*
Definition										
Type	Remote Displacement				Force					
X Component	0.01. m (ramped)						0. N (ramped)			
Y Component	0.01. m (ramped)						14199 N (ramped)			
Z Component	0. mm (ramped)				1.4775e+05 N		0. N (ramped)			
Rotation X	0.1 ° (ramped)									
Rotation Y	0. ° (ramped)									
Rotation Z	0. ° (ramped)									
Suppressed	No									
Behavior	Deformable									
Rotation X		0.1 ° (ramped)								
Rotation Y		0. ° (ramped)								
Rotation Z		0. ° (ramped)								
Rotation X			0.1 ° (ramped)							
Rotation Y			0. ° (ramped)							
Rotation Z			0. ° (ramped)							
Rotation X				0.1 ° (ramped)						
Rotation Y				0. ° (ramped)						
Rotation Z				0. ° (ramped)						
Define By					Vector					
Magnitude										
Advanced										
Pinball Region	All									

Case 5 - Longitudinal loads – Results – Composite Frame;

Total deformation:

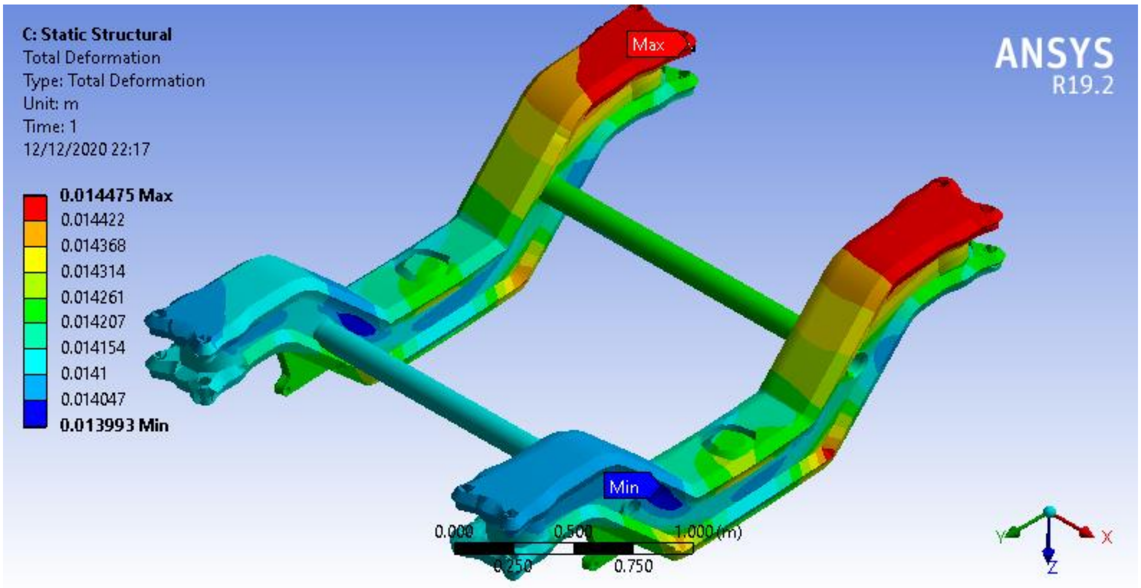


Figure 93. Total deformation 1 - CFRP - case 5.

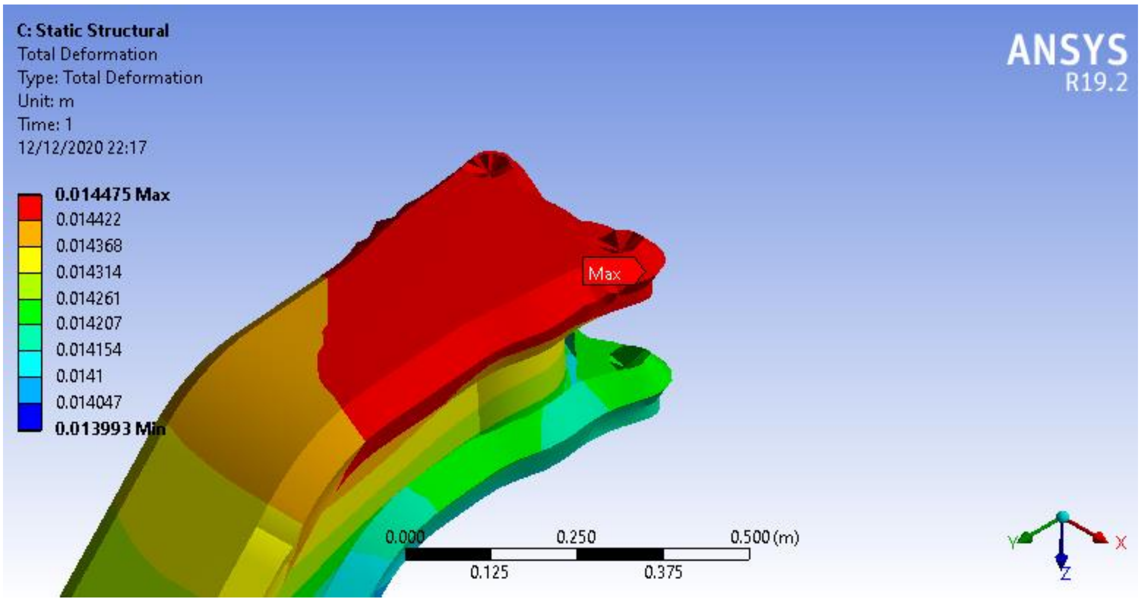


Figure 94. Total deformation 2 - CFRP - case 5.

Von-Misses stress:

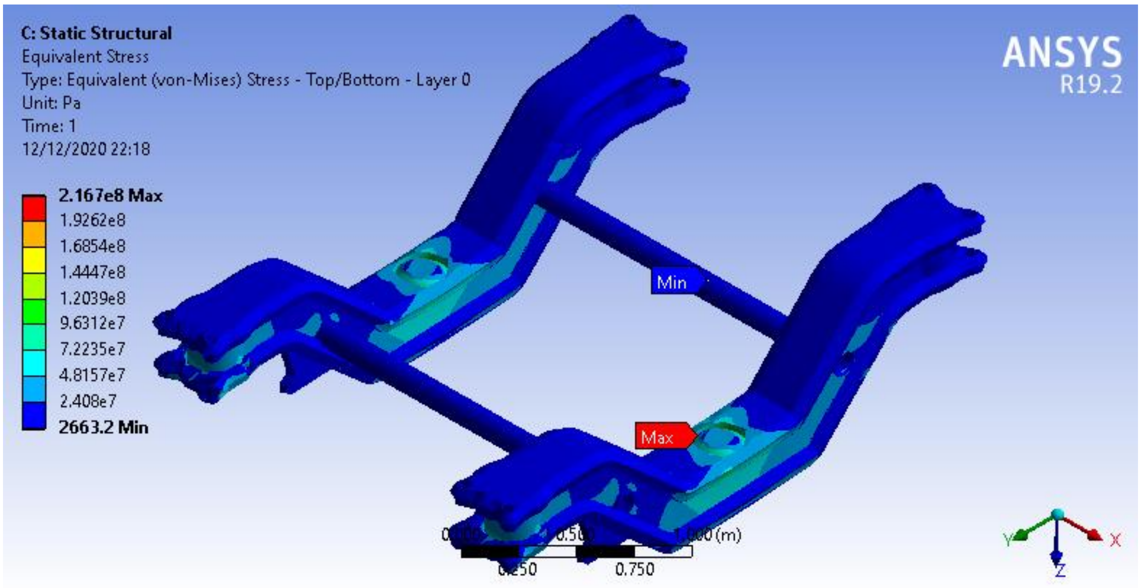


Figure 95. Von-misses stress 1 - CFRP - case 5.

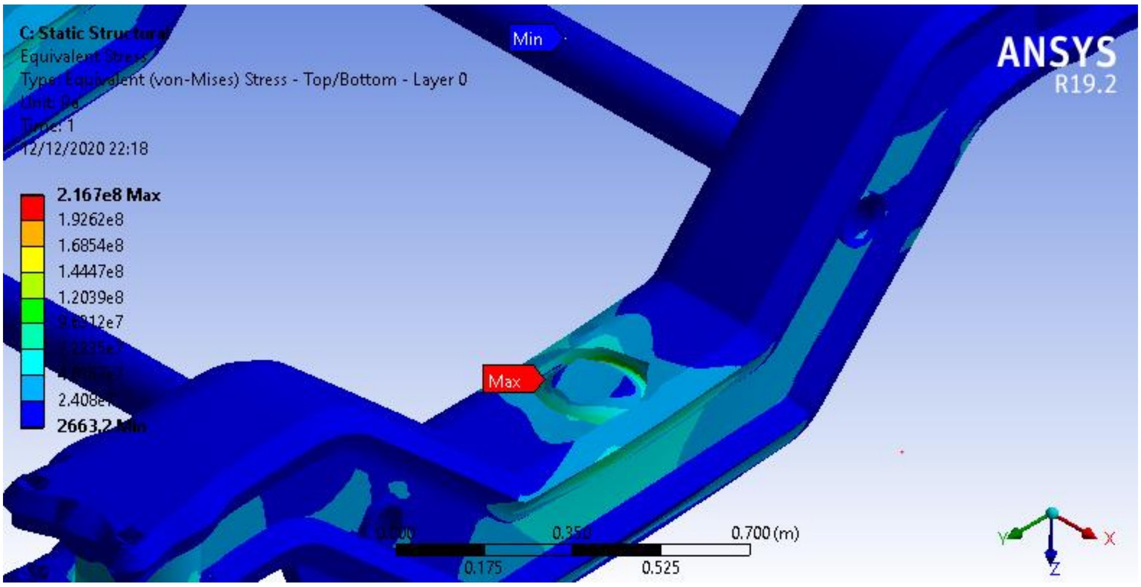


Figure 96. Von-misses stress 2 - CFRP - case 5.

Inverse fail factor:

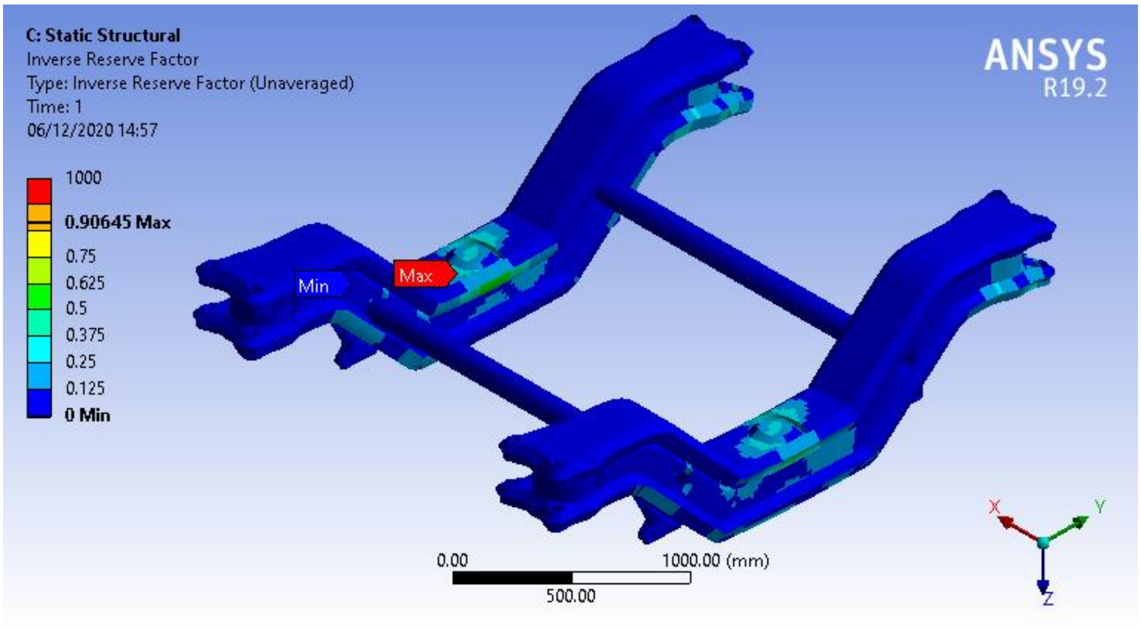


Figure 97. Inverse fail factor 1 - CFRP - case 5.

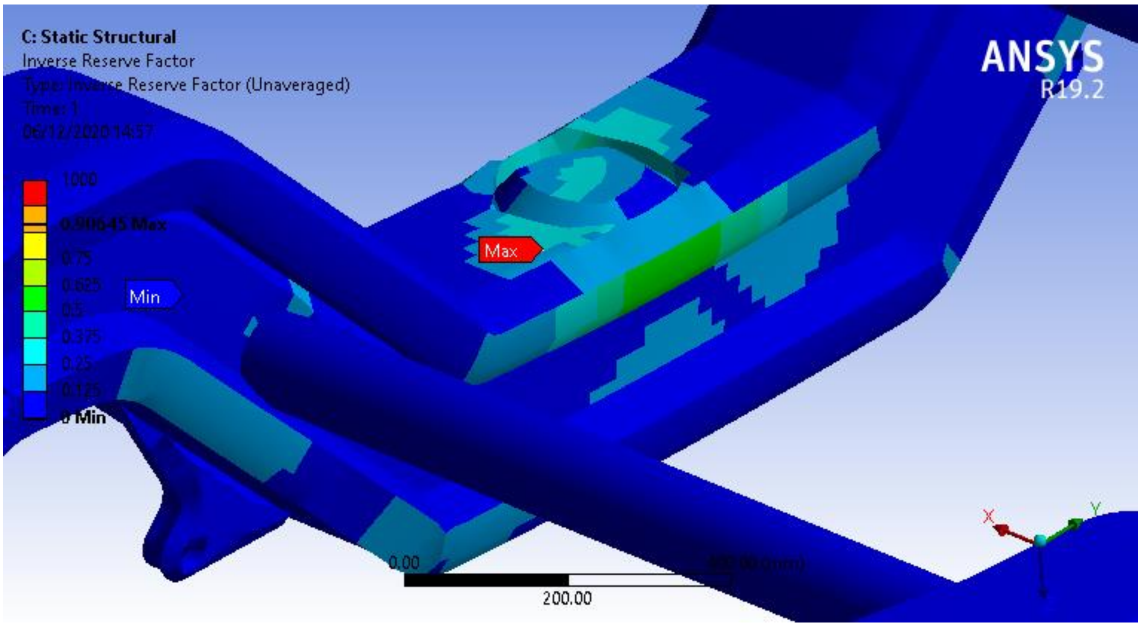


Figure 98. Inverse fail factor 2 - CFRP - case 5.

Safety Factor:

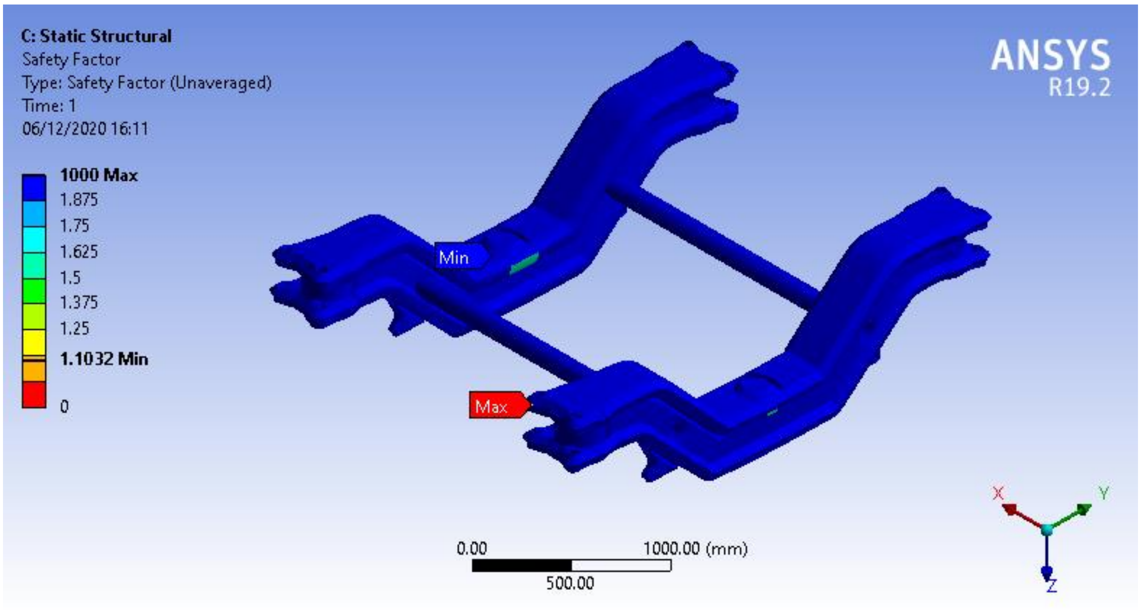


Figure 99. Safety factor - CFRP - case 5.

b. Appendix II– Steps needed in ANSYS to simulate both bogie frames.

In appendix II we will go through all the steps that was needed to achieve the simulations for both the structure steel S235 and the composite bogie frames.

That being said, the starting bogie is the Structure Steel S235 bogie frame.

Structure Steel S235 bogie frame:

The first bogie frame that was designed and simulated was the structure steel S235 bogie frame, out of both of the bogie frame, this one is considered the least complex and more straight foreward.

Like mentioned previously in the memory, the bogie frame was initially designed and assembled using SolidWorks, later imported into ANSYS R19.2 using the IGES format. This format allows the models to be imported into ansys with SOLID geometry, great to start simulating.

The first step to usig ANSYS is to oopen the *Static Structural Analysis*, for this you would need to go into the left tree of tools in the main ANSYS interface and select the *Static Structural Analysis.*

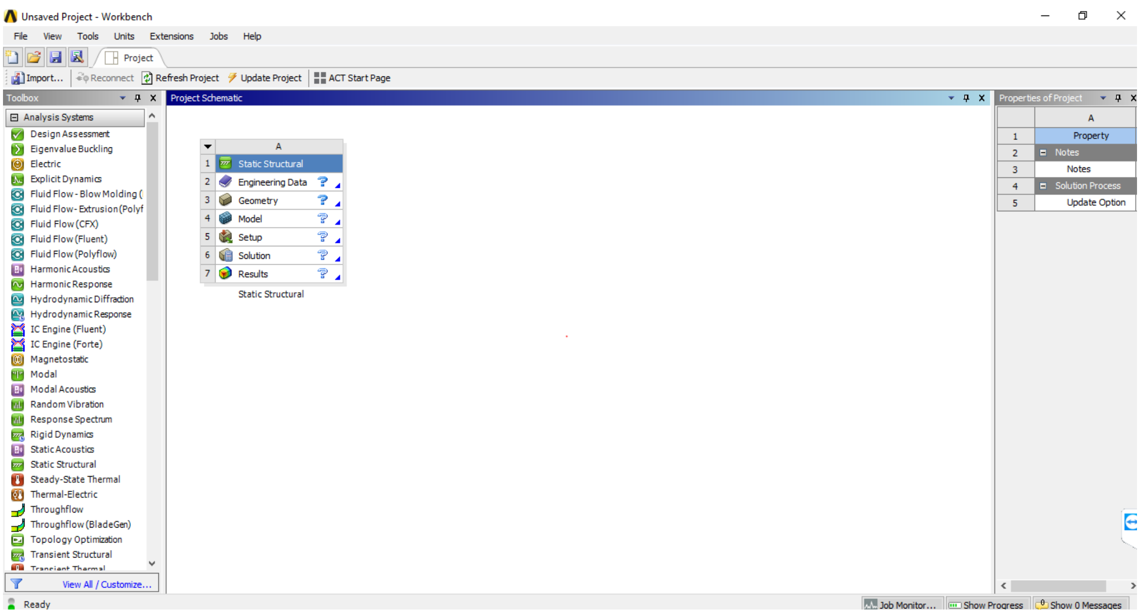


Figure 100. Screenshot of the main ANSYS interface with the Static Structural Analysis enabled.

Once you have selected the analysis you need to do the simulation, you would need to select the material that the model will have and for that you need to go into the Engineering Data that is visible in the box titled *A*.

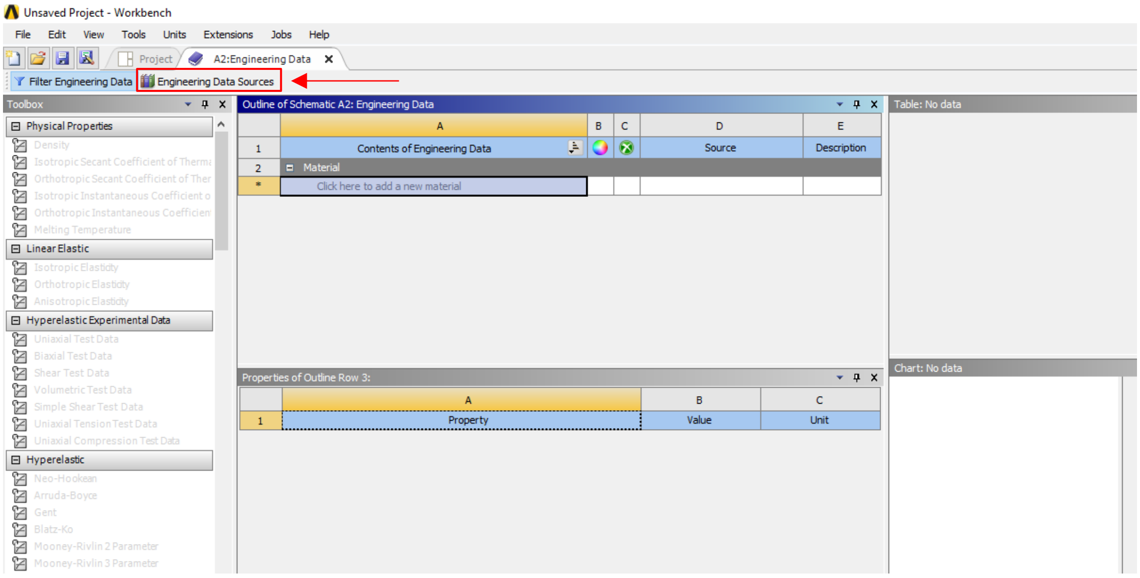


Figure 101. Engineering data interface that is needed to select the materials present in the model.

Now that you are in the *Engineering Data*, you need to select the material by accessing the *Engineering Data Sources* on the top left of the screen (see red arrow and box in the figure above). Like we are first describing the steps for the structure steel s235, we will go and look for the structure steel material in the data source.

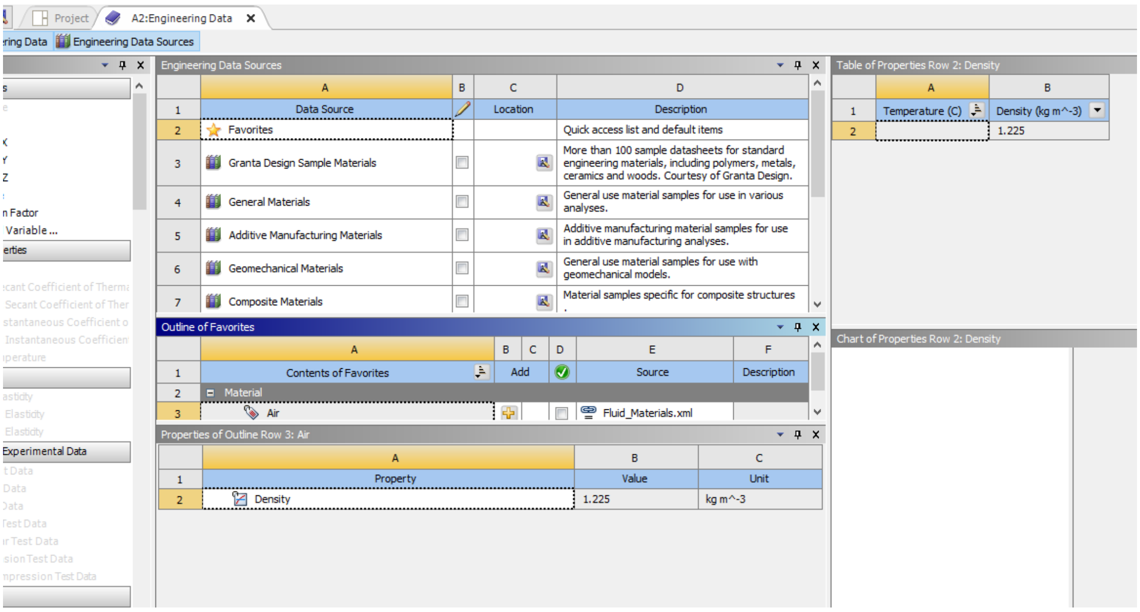


Figure 102. Information after clicking engineering data source.

Like mentioned previously in the memory, the properties given by the software ANSYS concerning the materials were detailed and good enough to use, that being said, to make the material fit into the narrative of using *Structure Steel S235* the main properties for the structure steel will be modified so that it simulates that of the *Structure Steel S235.*

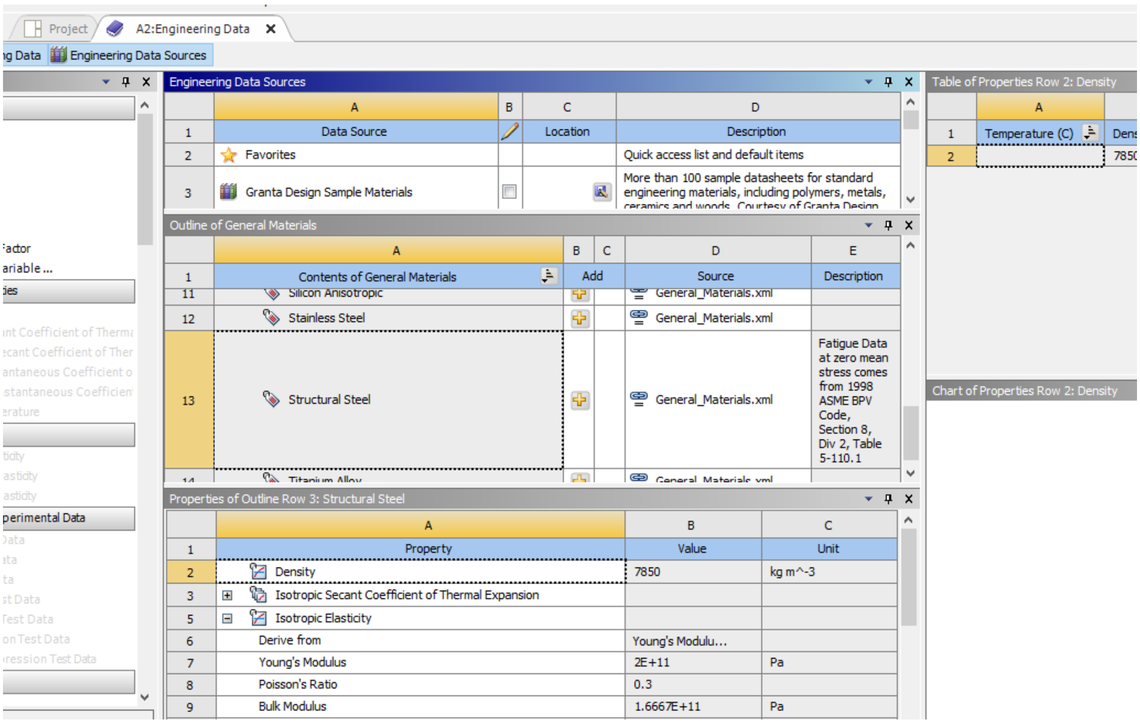
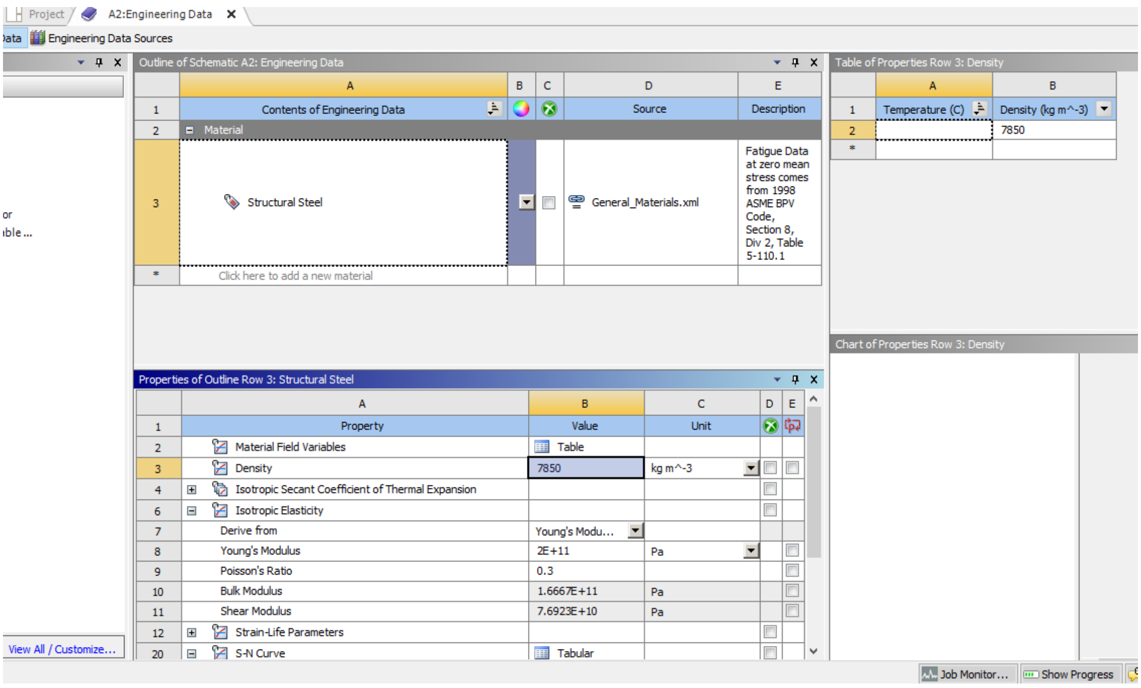


Figure 103. Search for the structural steel in the General material section.

To be able to select the structural steel you would need to click in the engineering data source tap General materials, and as you do so, a list of materials will pop-up in the bottom tap. In this list you will find the *Structural Steel* and by clicking the plus sign you will add it to the materials selected that will later on will be used in the simulation. It is worth mentioning that in the bottom tap you can see that the properties that the structure steel, of which ahead will be modified.



Once you select the material you leave *Engineering Data Source* by clicking once again in the top left bottom and it will exit you to the engineering data main interface. In this interface you can see that the *Structural Steel* was selected, and, in the bottom, you can notice that the properties are listed. These properties will be modified to that of the *Structure Steel S235*, in the appendix III is available the technical data sheet for the *Structure Steel S235.*

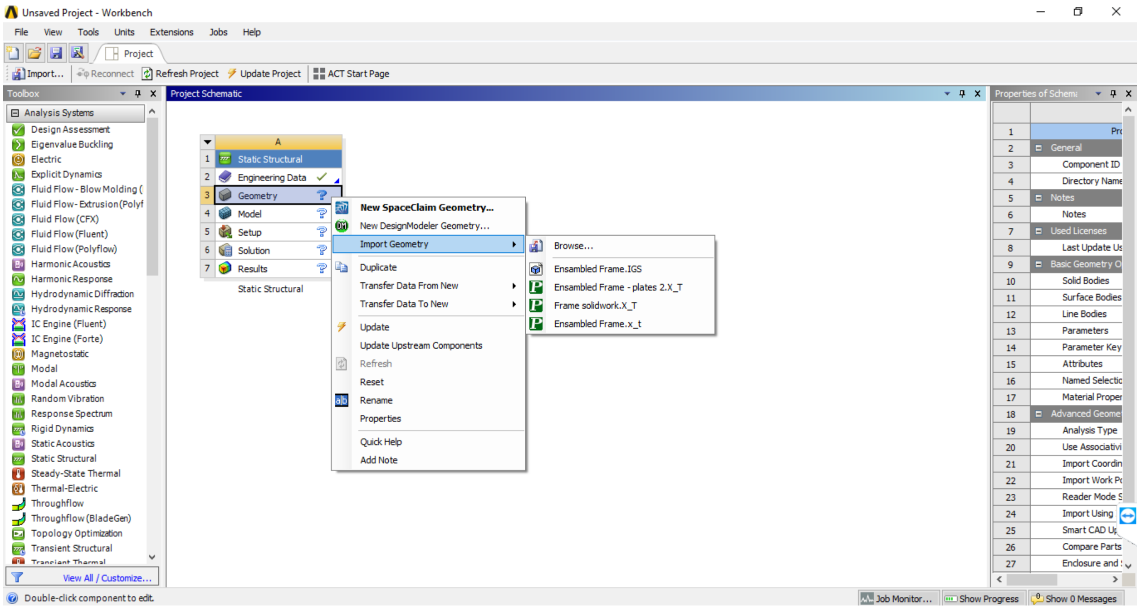


Figure 105. Importing the model.

The next step is to import the model, to do that you would need to right click on the the *Geometry>Import Geometry>Ensabled Frame.IGS*, remember that the imported models sometimes has some errors. These errors can arise when switching from one software to another and is convenient to deal with them at this stage of the simulation to safe headaches when trying to search for a convergence.

To do this you need to go into geometry and do a few analysis to the model. To access the geometry interface (called SpaceClaim) you need to doble-click on the geometry tap and the interface will open up.

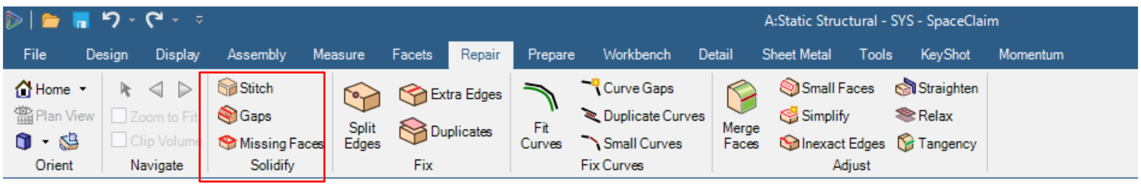


Figure 106. Repair bar on top of the SpaceClaim interface.

Once you are in the SpaceClaim interface you would need to do some analysis on the model, as stated before the model sometimes imports itself with some underlying errors so you must fix them prior to any simulation. For this, you will go to the repair option (like in the figure on top) on the top of the screen and use all 3 available tools, stitch, gaps, and missing faces to fix the model in case that there is a need for it. Ideally there will be no need, but there is always room to dedicate a bit of time to fixing the model to prevent headaches ahead.

After this, we will go over how to simulate the model using the ANSYS mechanical. To access it, you will need to go back into the ANSYS main interface from before and doble click the figure.

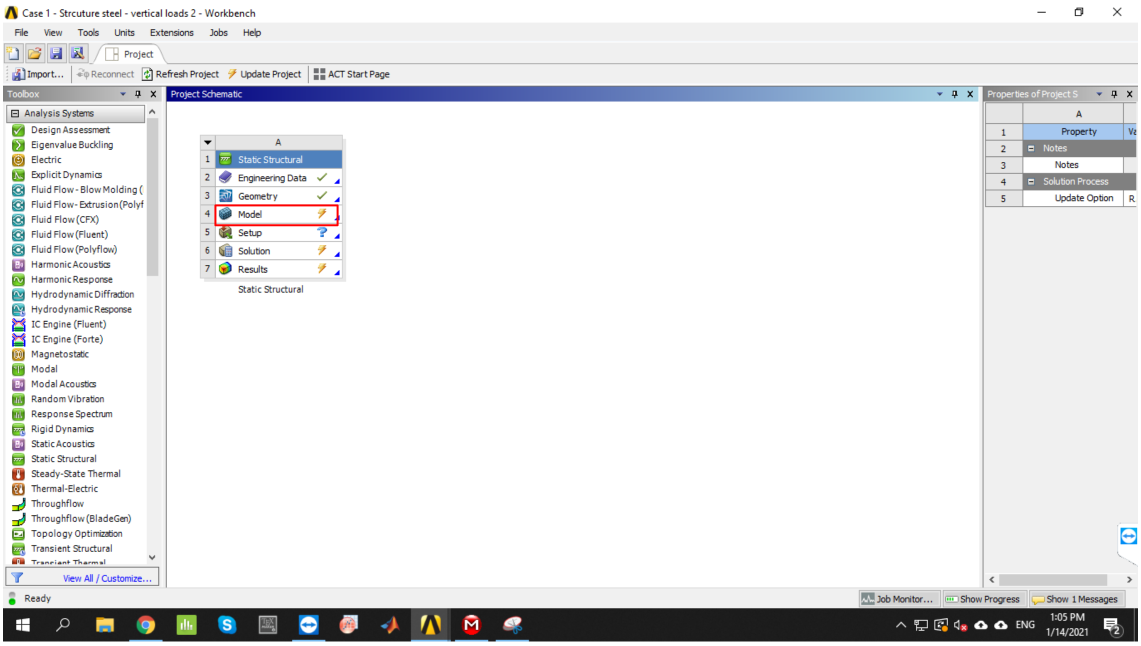


Figure 107. Main ANSYS interface.

from here on we will have to select a case study to explain the procedure, for this the study case that there has been selected is the case 5. This case is the more complete and has the majority of the forces that will be needed for the simulation.

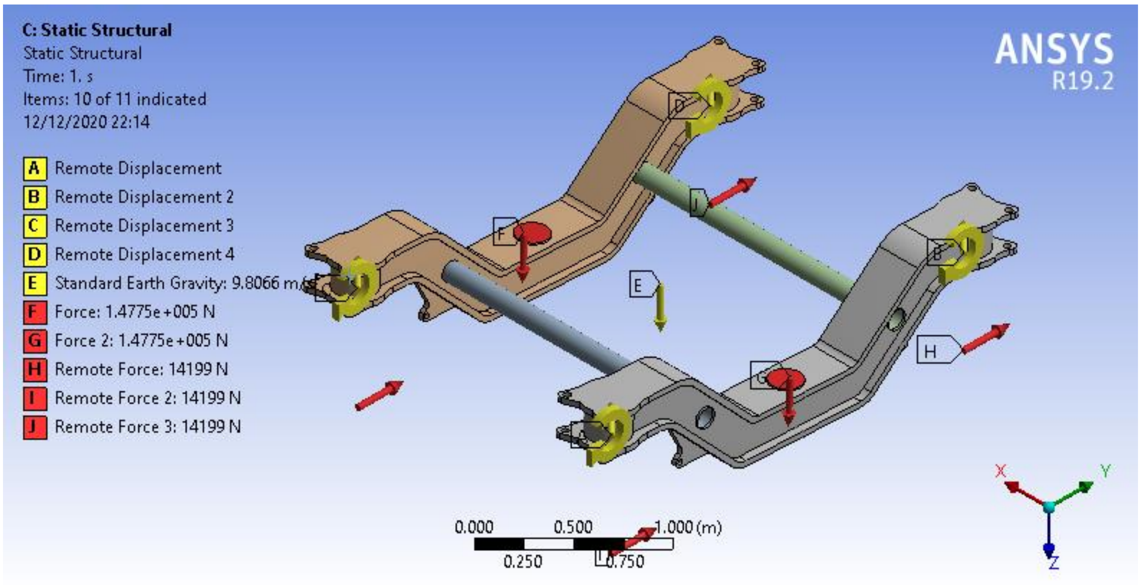


Figure 109. Case 5, isometric view to see the forces and the restraints.

Above is the figure of what we are aiming for, for this there are some previous steps that will be needed to take prior to start the simulations. The first thing that ANSYS mechanical needs from you is to declare the material for the model.

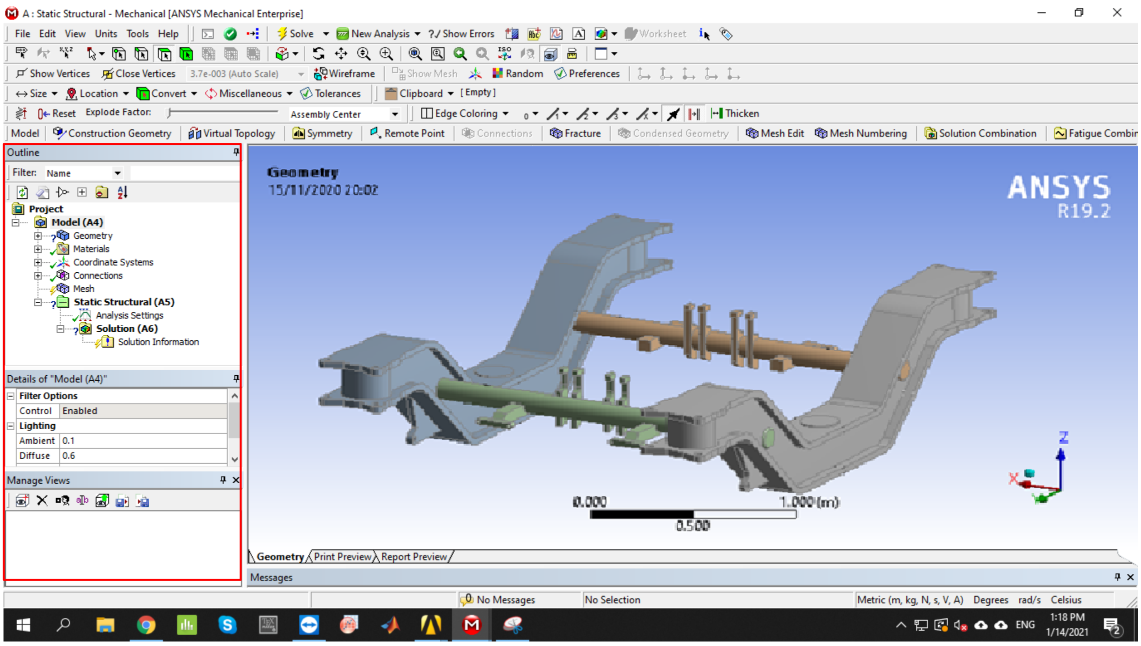


Figure 108. overview of the ANSYS mechanical.

On the left side of the screen is the model tree that will be used to simulate the model. This tree has all the tools necessary so that you can accurately achieve the goal. The interrogation marks that the tree has means that there is a need of an input to go ahead, as to that the first step is to choose the material of the geometry.

By clicking on geometry on the tree of tools, on the bottom top that is directly underneath appears a yellow highlighted box. This box will need your input to go ahead. The box asks you about the material of the part of the model you have selected, but because the model is all completely structural steel S235, then the fasted approach would be selecting everything at once and choosing the material to be structural steel. After doing so, you will have the following view of the model.

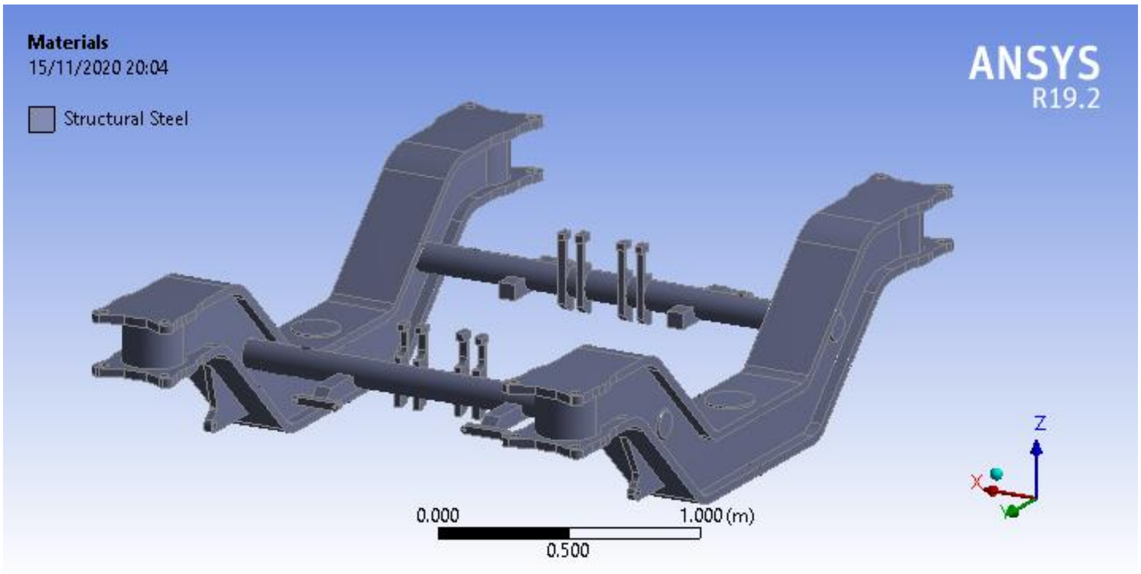


Figure 110. In grey the structure steel parts of the model.

Once the model has been correctly materialized, the next step would be the analysis of the connectors. Like mentioned previously, the model sometimes imports with some penetrations and errors, so when you do not fix the model, you may get some areas where there is too much pivoting (or freedom in a certain node of the model. To combat this, we previously passed the model in a repair face in SpaceClaim to avoid these problems. The proper way of knowing if you have a penetration or gap issue in the model is to go on the Contact on the tool tree, right-click, and select contact tool. Once there, you go on the Contact Tool, and right-click again to add the *status* option, once this is done you can initialize the contact analysis and the following figure is what you should have displayed on your screen.

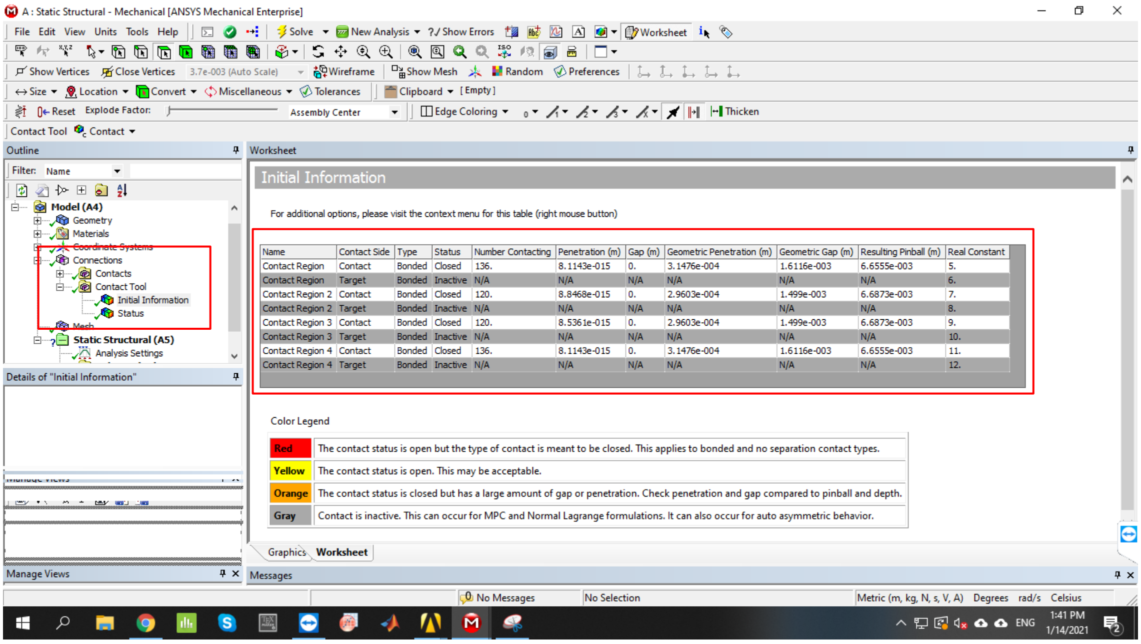


Figure 111. Initial information of the status of the contact point and nodes on the model.

As you can see there is no red, orange, or yellow warning signs that ANSYS is displaying, guaranteeing that the connections and results that will be given are accurate and reliable.

The next step of the simulation is to do a mesh of the model, and the according meshing quality analysis for the model, this step will not be revised due to it being already gone over in the memory (C*hapter. 9 ANSYS simulations*).

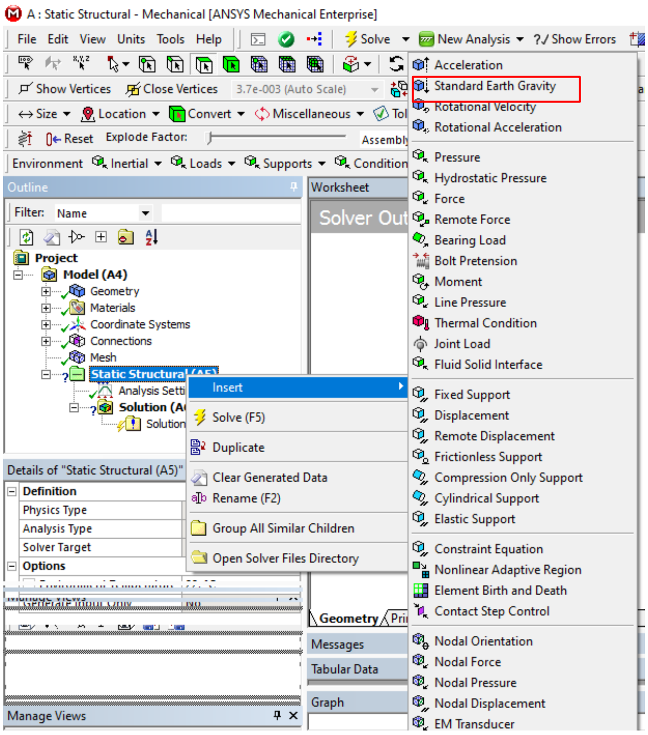


Figure 112. Standard earth gravity.

The subsequent step after the meshing of the model is adding the forces and restrains that the model needs to fulfill the simulation of case 5. To do this, you will need to go on the tool tree on the left-hand side of the screen and right click on static structure. There will be many different options of forces that appear when doing this. Like you may notice, there are 3 different style of forces that are present in the model, starting with the gravitational force of the model itself. This force is the yellow force that appears in the figure above where you can see all the constraints and forces for the case 5 scenario. To add this force to the simulation, *tool tree> static structure> Insert> standard earth gravity*.

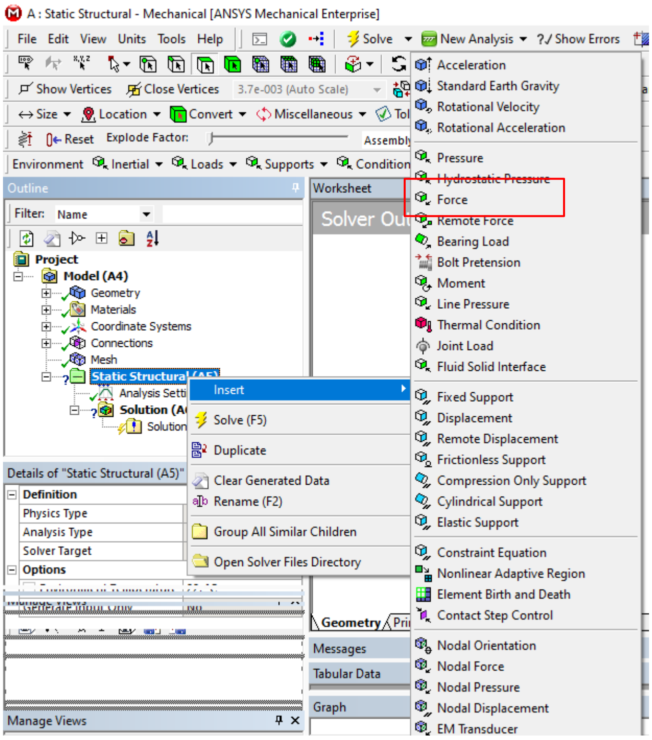


Figure 113. Insertion of the vertical forces.

The next force that is going to be added are the vertical forces, these forces are the forces transmitted by the secondary suspensions on top. These forces could be simulated by a spring force over an area, but it has been decided to simulate it in a way that the force goes from 0N to the goal in newtons gradually and over a surface that has been bumped upwards, to simulate the area of initial contact of the spring. This is a valid representation and an easier alternative to simulating a spring in ANSYS mechanical. Thus, to do this, you need to access the *tool tree> static structure> Insert> Force.* As you also may have noticed there are two vertical forces, and as such you would need to do this process two time, one for every vertical force. When adding the option force into the tool tree, on the bottom tap immediately under, you have to input the magnitude of the force and where this force will be located (you need to select the area where the force will be connecting with the model). In the memory of this project, it has been calculated that the vertical force will be of a magnitude of F = - 134388 N for the Z-axis. To do so, you must go into the option *Defined By> Components*, so you can have a X, Y, Z axis available. There in the Z-axis box, you input the value of the vertical force stipulated in the memory. You repeat this procedure for the other vertical force.

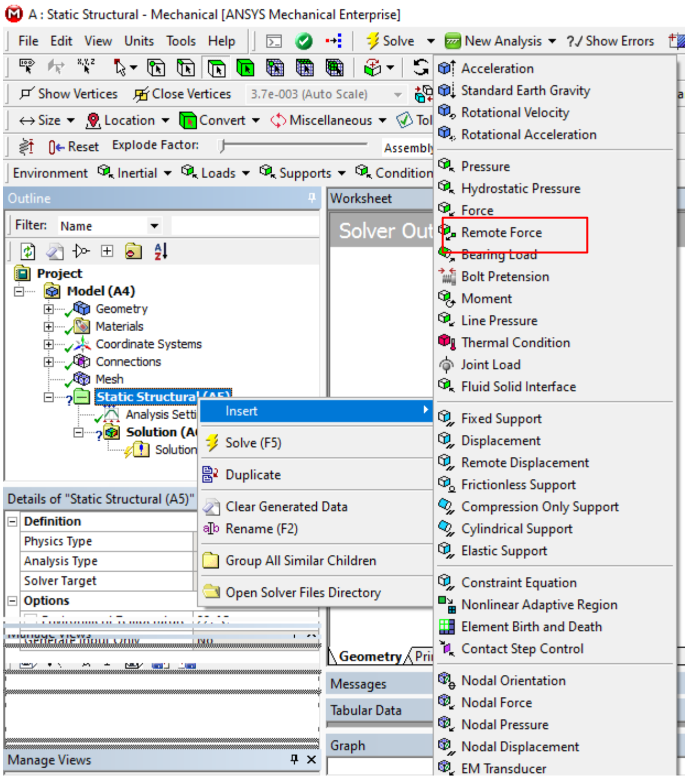


Figure 114. Insertion of the longitudinal remote force.

Lasty concerning the forces, we will need to input the remote forces on the bottom side of the frame replicating the forces that appear from the contact of the rails and the wheels of the train. As we are studying case 5, Lozenging forces, these are the longitudinal remote forces start their influence when braking and it has a magnitude of F = 16628 N that is calculated in the memory of this project. Like you can see I the overview of case 5, there are 4 remote forces. These forces all have the same value and the same direction. Just like the previous forces, the way to activate these forces on the model is by tool tree> static structure> Insert> Remote Force. The difference from the force option, is that the remote force is used when the force is in a certain distance from the face, and just like we mentioned just, the forces that they simulate is that of the braking of the train and the contact point is where the wheels meet the rails.

Now, we will proceed onto the constraints. The constraints on the model will be following the EN 13749:2011 standard as a guideline. The guideline mentions that there is a need for a level of freedom when supporting the frame, and as such, the selected amount of freedom follows the table ahead.

Table 14. List of the restrictions for all the cases in the study of the composite bogie frame.

	Support 1			Support 2			Support 3			Support 4		
Cases ↓	x [mm]	y [mm]	z [mm]	x [mm]	y [mm]	z [mm]	x [mm]	y [mm]	z [mm]	x [mm]	y [mm]	z [mm]
1 2	10	10	0	10	10	0	10	10	0	10	10	0
3	10	10	0	10	10	10	10	10	0	10	10	10
4 5	10	10	0	10	10	0	10	10	0	10	10	0
Cases ↓	Rx [°]	Ry [°]	Rz [°]	Rx [°]	Ry [°]	Rz [°]	Rx [°]	Ry [°]	Rz [°]	Rx [°]	Ry [°]	Rz [°]
1 2	0.1	0	0	0.1	0	0	0.1	0	0	0.1	0	0
3	0.1	0	0	0.1	0.1	0.1	0.1	0	0	0.1	0.1	0.1
4 5	0.1	0	0	0.1	0	0	0.1	0	0	0.1	0	0

There is an average 10mm amount of freedom to simplify the constrains. You can see that for case 5, there is horizontal and longitudinal freedom of 10 mm, but 0 mm of freedom vertically. Later, in the rotational aspect, there is 0.1 degrees of rational freedom for only the X-axis and the so that it can rotate longitudinally. Case 3 is the only case that does not follow this order, but it is because this case is the unloading scenario, where the forces are bit different and the restrictions as well.

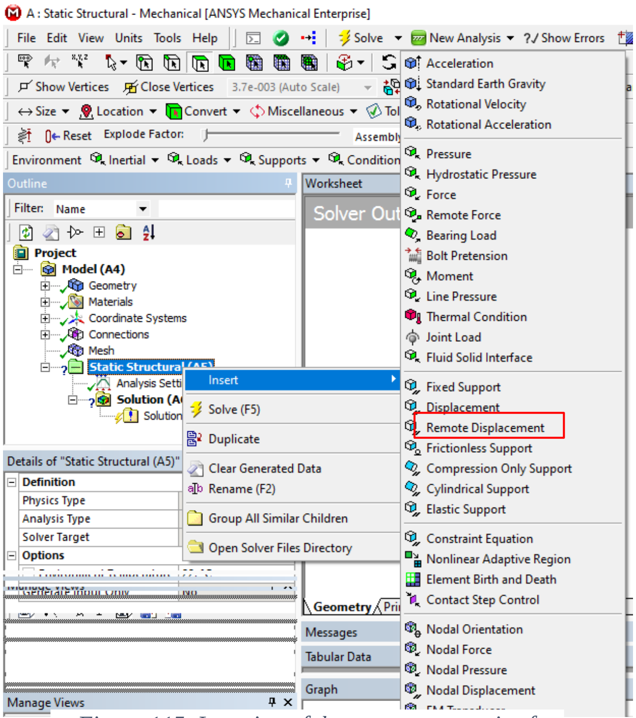


Figure 115. Insertion of the remote constrains for the bottom part of the frame.

The method to including these restrictions to the model is very similar to the forces, you would initially need to go to *tool tree> static structure> Insert> Remote Displacement.* After doing so, you will be asked the amount of liberty you would want to give to the model (i.e., the liberty of the X, Y, and Z components and the Rotation of X, Y, and Z). Select the faces, input the amount of freedom, and repeat 3 other more times until you reach a total of 4 times.

Once that you have finalized the forces and constrains present in the simulation you will need to insert which are the solutions that you are looking for in the simulation, as this is a tension simulation, we will be looking for the Total deformation, Von-Misses stress, and the safety factor.

To add the all the solutions mentioned previously you need to go into the tool tree one more time,

Tool tree> Solutions> Insert> Deformation> Total

Tool tree> Solutions> Insert> Stress> Equivalent (Von-Misses stress)

Tool tree> Solutions> Insert> Stress Tool> Max Equivalent stress

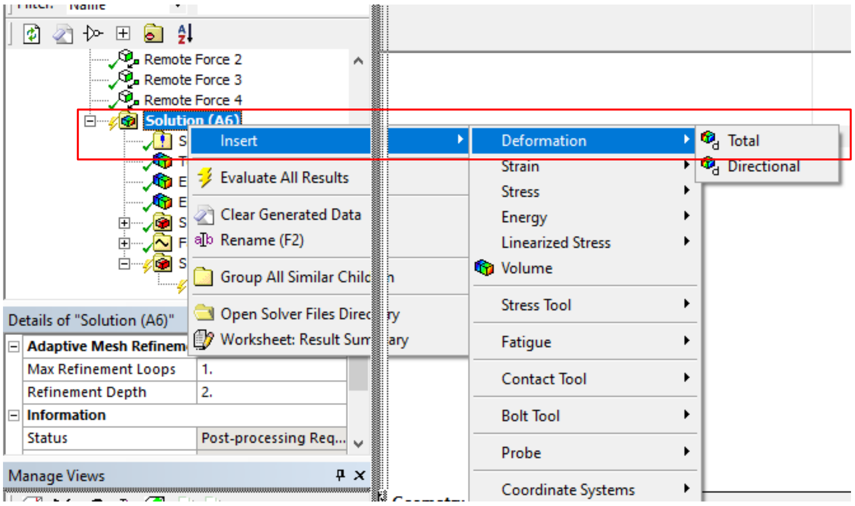


Figure 116. Solution selection - Total deformation.

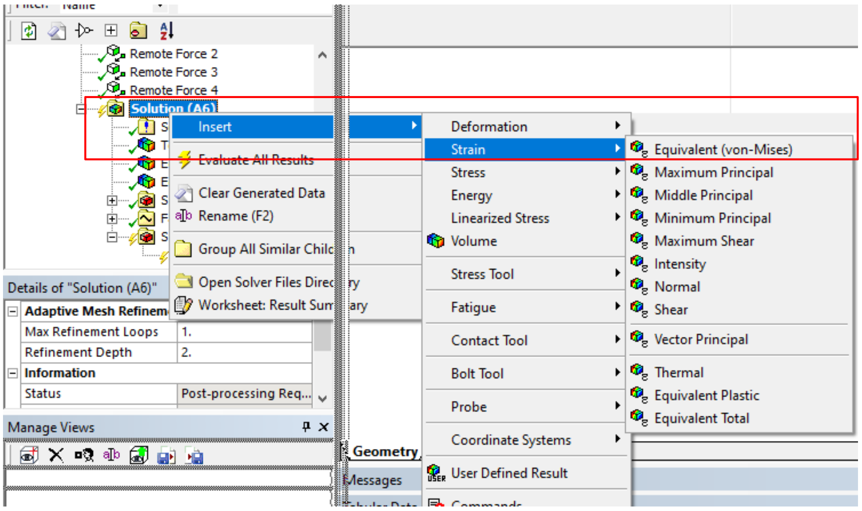


Figure 117. Solution selection - Von-Misses Stress

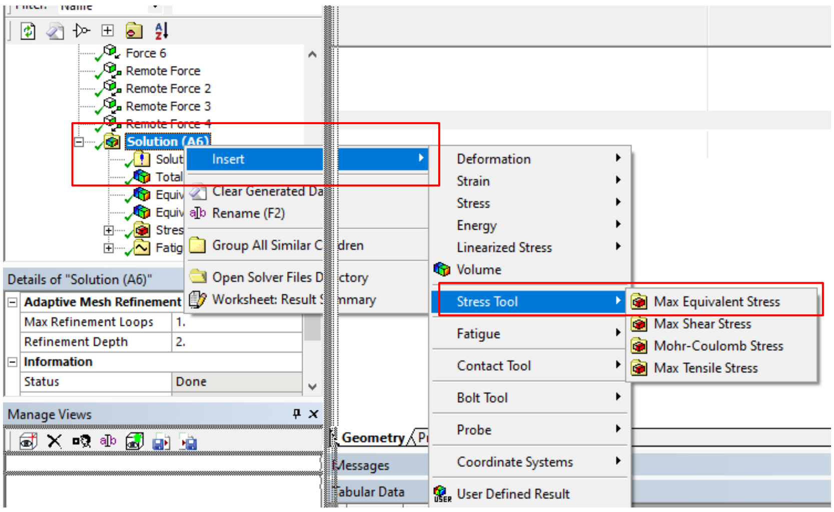


Figure 118. Solution selection - Safety factor

Above are screenshots that express visually the steps you need so that you can input the solutions that you are looking for. In this study, we were intersted in knowing the total deformation, the Von-Misses stress, and the safety factor of the model under the different scenerarios, to later put into comparison with the same scenearios but with the composite frame, and derive conclusions.

With this last step we can now run the simulation and understand the results.

Composite bogie frame:

This study is also a comparison with a composite frame version, and as such, we need to explain the procedures that was needed to go through to complete the explaination chapter. In the memory, you can find the ACP (Pre) and the Mesh (mesh quality) for the composite frame, and so it will not be explained twice.

To guarantee the fluent understanding of the composite ANSYS explanation, Case 5 will be choosen just as we choose it for the structure steel S235.

Staring off, the setup for the composite frame is the folowing.

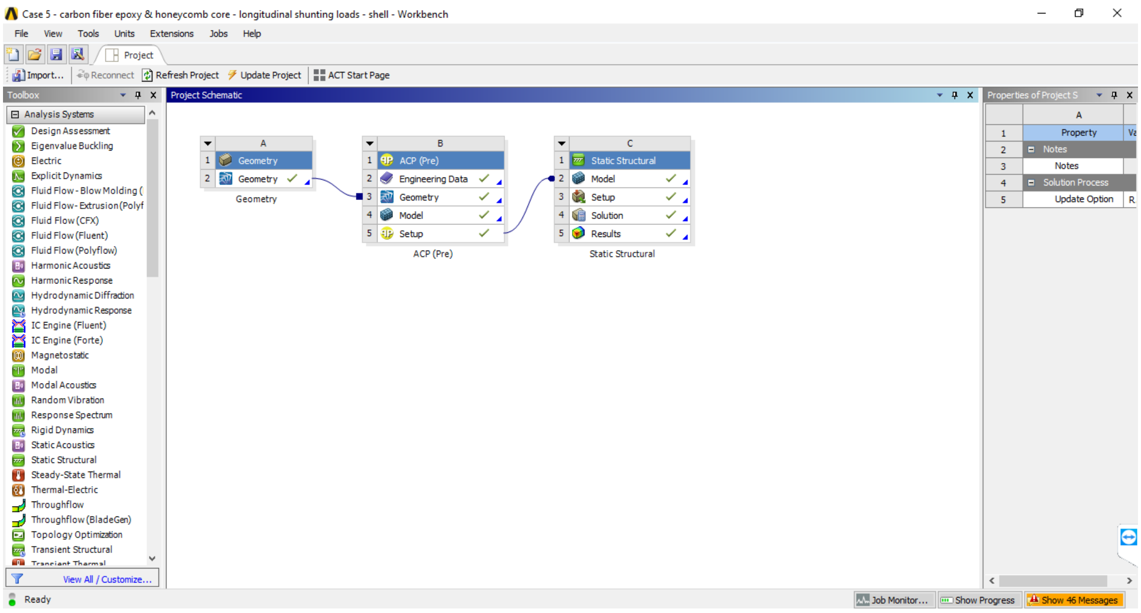


Figure 119. ANSYS main interface when finalized using it for the composite bogie frame.

In the above picture you can see a few differences with how the main interface looks, as a starters, the composite frame has linked 3 different modules. Starting off with the *Geometry*. The geometry module is the same as the SpaceClaim but it is seepareted, it has the same function as it did in the the static structure for the structural steel S235 but it is outside of the main module. Next is the ACP (Pre) module, this module is only needed when you are constructing or using the composite format available in ANSYS. It does not have any other application is not to shape and create the composite ply that you desire, and only works with SHELL geometry too. Lastly is the static structural, this operates the same way it operates with the structrual steel S235.

To start the explanation, you will need to import the model that you would like to trasform into composite. ANSYS is very powerful, it allows to work with composite materials and to do so, you must import the model as a SHELL geometry model. By doing so, you can use the ACP (Pre) module that you will need to create the composite bogie frame.

Just like in the previos case, you will need to use SpaceClaim to clean and repair the model, so that you do not have any error ahead. This step is very important specially when using ACP (Pre), and is always convenient to take into consideration when designing anything.

Next you must go into the material selection aspect. Just like in the previous case, we access the materials that ANSYS has available by going into the *Engineering Data*. For this you must first add the ACP (Pre) into the main ANSYS interface, in the Component Systems you can locate the ACP (Pre) and you can add it by doble-click on it.

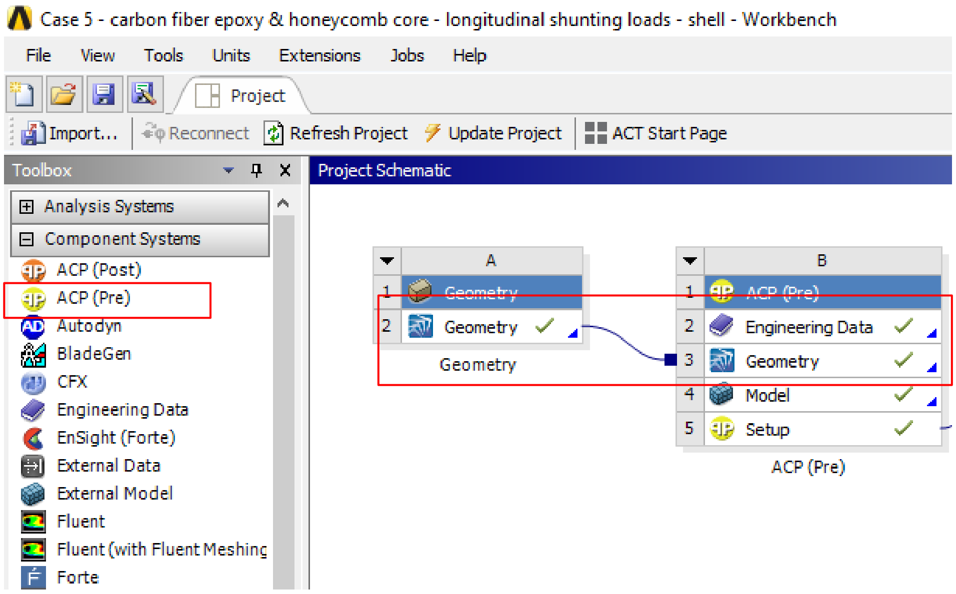


Figure 120. Selection of the ACP (Pre) module.

You can link both modules by dragging the geometry module into the ACP (Pre) one, impporting that way the SpaceClaim repaired version of the model into the ACP (Pre) module. Afterwards,the next step is selecting the composite materials, in the study we have selected the existance of the Recycled Unidirectional Epoxy Carbon Fiber, Waved Epoxy Carbon Fiber, and the Aluminum Honeycomb. For that, just like in the structure steel S235 you need to go into the *Engineering Data> Engineering Data Source*

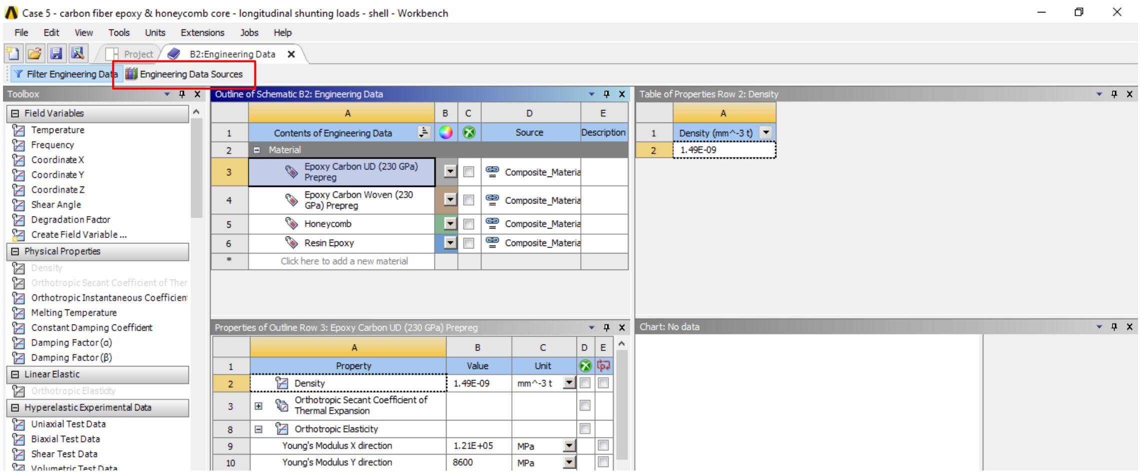


Figure 121. Engineering Data interface.

Inside the *Engineering Data* source you will need to seach for the composites, the composite group of materials are located in the *Composite materials*, once in there is a selection of 23 composite materials that are available to use and modify. In this list are the materials that will be present in the composite model. Select them.

Once all the three materials are selected, to accurately simulate the recycled version of the unidirectional epoxy carbon, we needed to change the properties according to the study found referenced in the memory document.

After selecting the material and repairing the model, the ACP (Pre) module will be needed to be initiated, and to see the developemnt, refer to *chapter. 9.2.1 ACP (Pre) module* in the memory document. The ACP (Pre) module is needed to construct the composite plies, to select where and which are the directions of the primary orientations for the composites.

Lastly, you need to link the ACP (Pre) module after the utilization of it with the *Static Structural* by connecting the Setup boc in the AC (Pre) with the model box on the *Static Structural.*

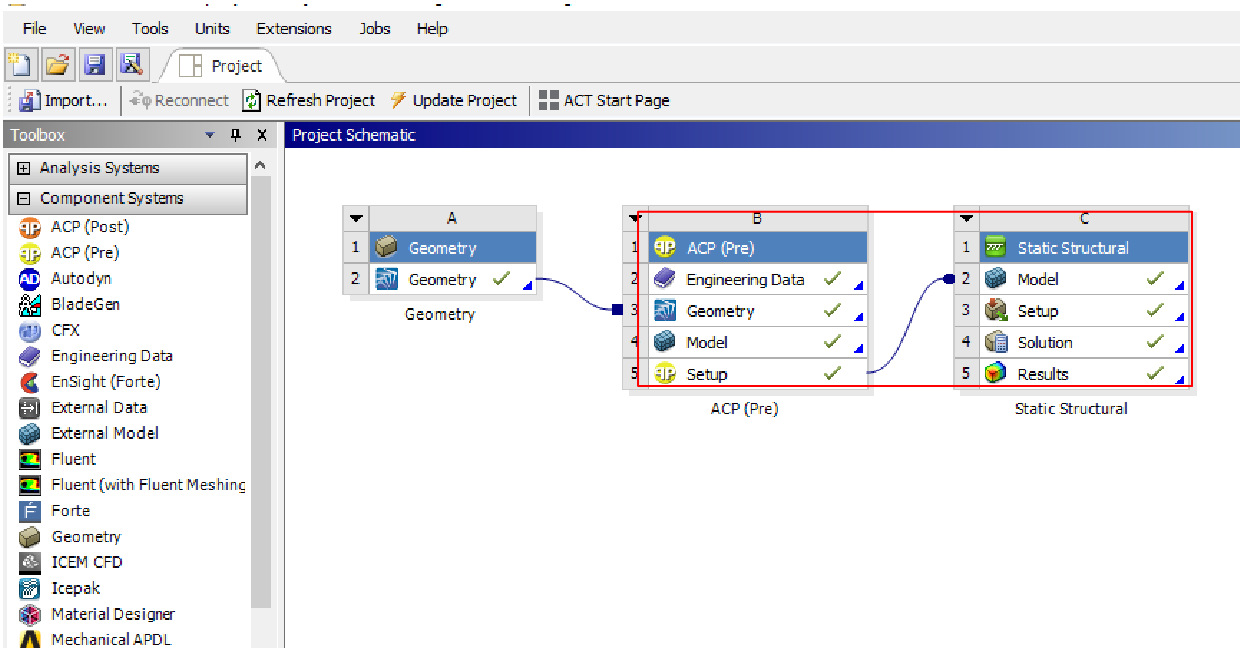


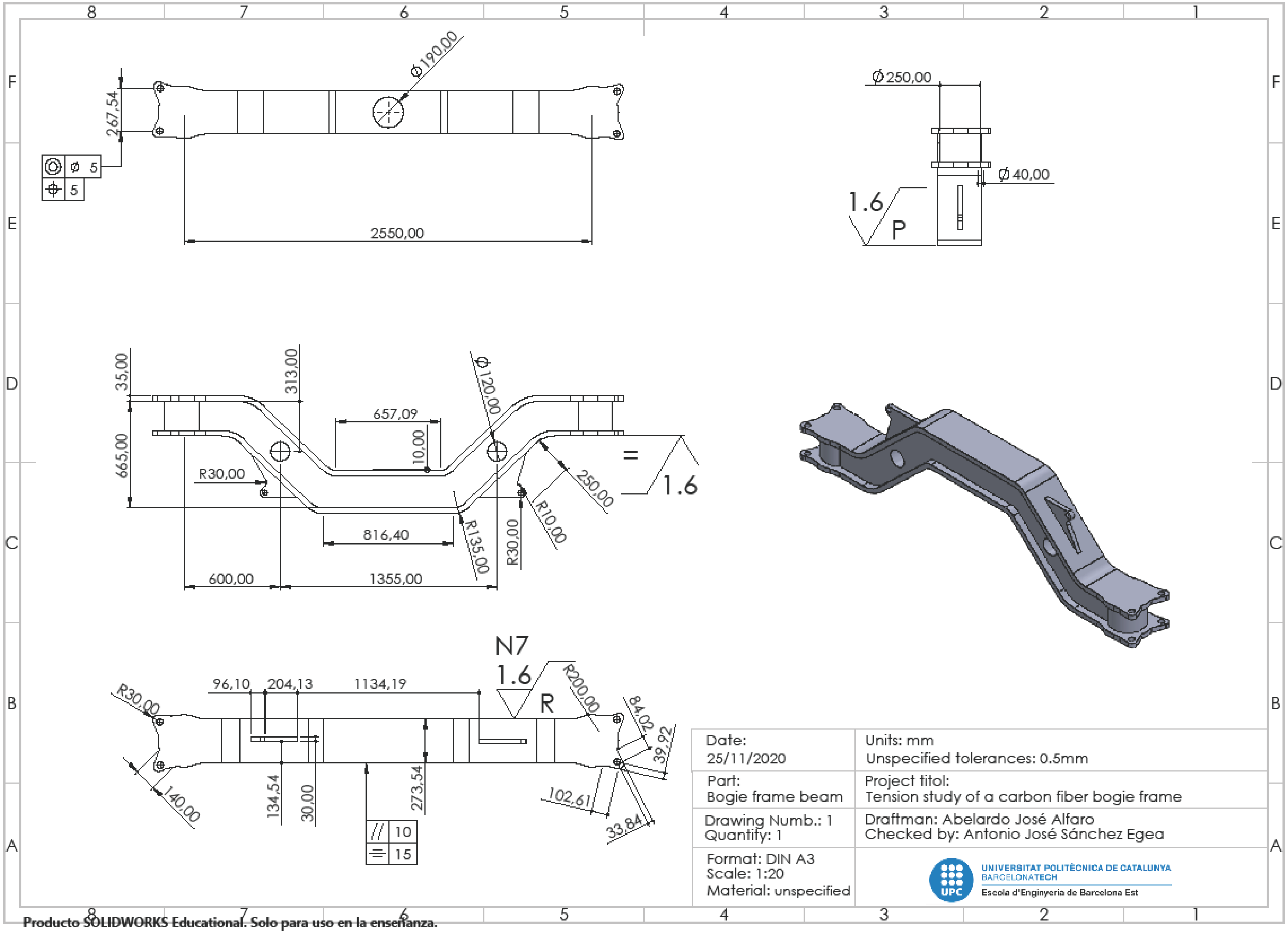
Figure 122. Connecting the ACP (Pre) module with the Static Structural module.

The steps inside of the static structural module are exactly the same as the ones explained previously but the difference is that the Mesh is not the same as for the Structural steel S235 model, this being said, the mesh is shared with the other 5 cases that the composite bogie frame is put under. The explanation and demostration of the mesh formation is also explained in the memory document, *chapter. 9.3 Meshing the composite frame.*

c. Appendix III – Technical drawings:

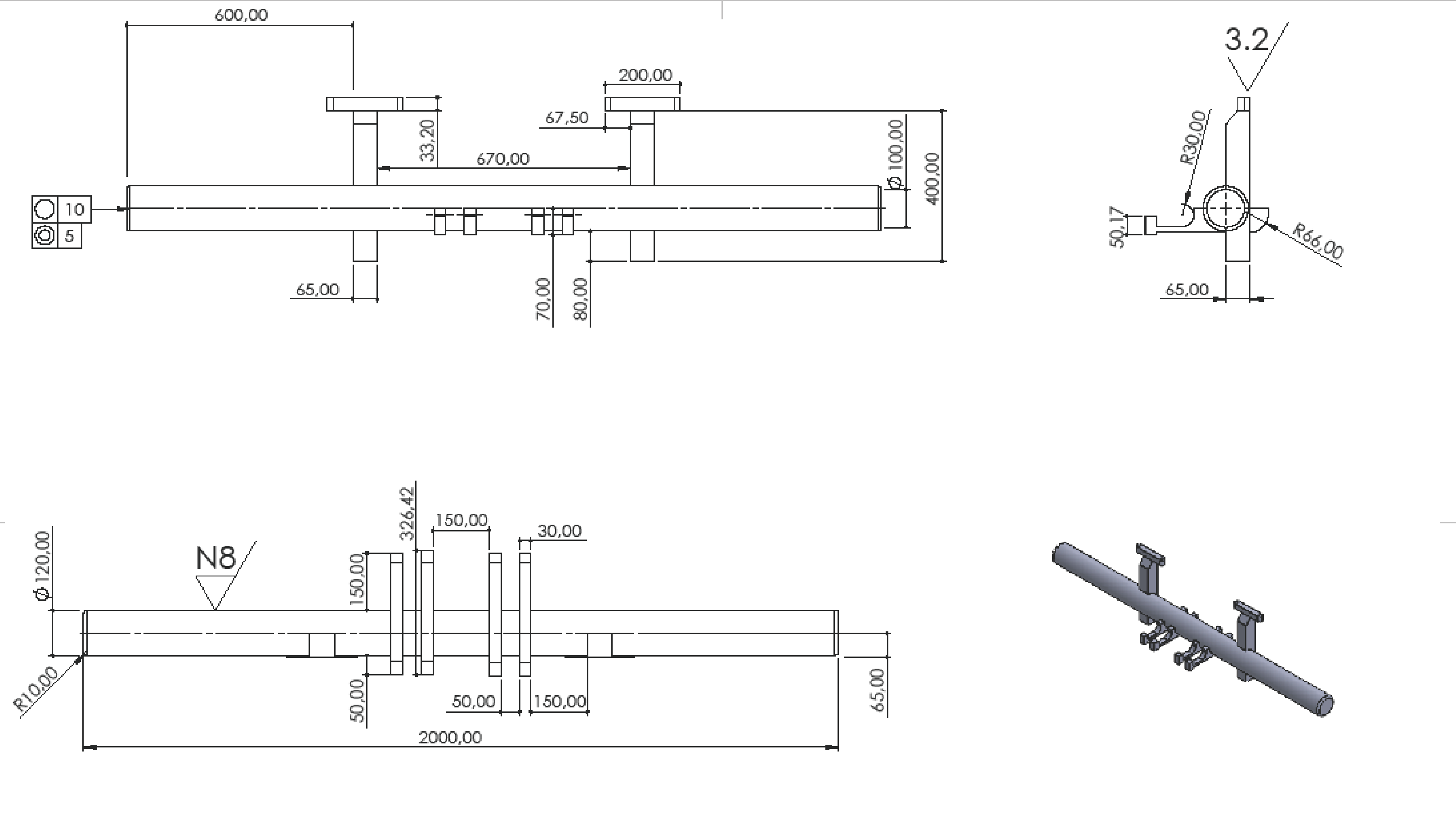
In this second section of the appendix, there is going to be a display all the technical drawings created after the designing both the composite and standard bogie frame. The technical drawings are going to follow the following order: Beams of both the composite and structure steel s235 bogie frame, connecting rod for composite bogie frame, connecting rod for the structure steel s235 bogie frame, assembled structure steel s235 bogie frame, and finally assembled composite bogie frame.

UNIVERSITAT POLITÈCNICA DE CATALUNYA
BARCELONATECH
Escola d'Enginyeria de Barcelona Est

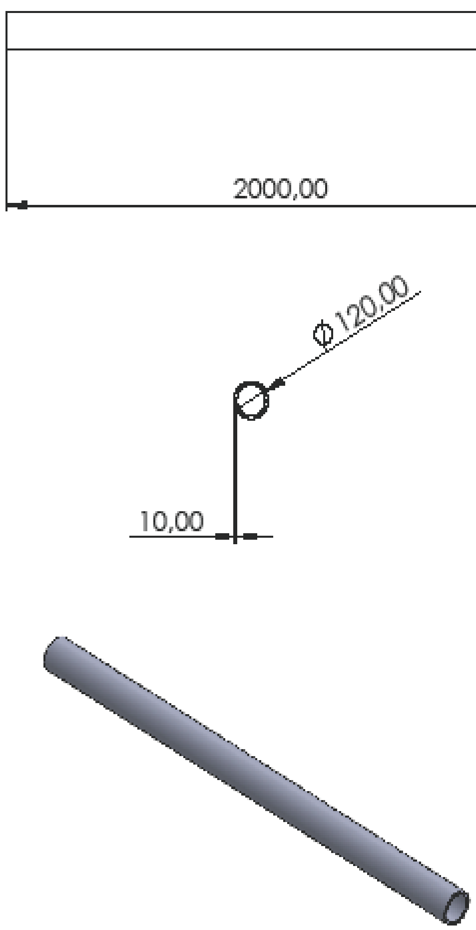


Date: 25/11/2020	Units: mm Unspecified tolerances: 0.5mm
Part: Bogie frame beam	Project titol: Tension study of a carbon fiber bogie frame
Drawing Numb.: 1 Quantity: 1	Draftman: Abelardo José Alfaro Checked by: Antonio José Sánchez Egea
Format: DIN A3 Scale: 1:20 Material: unspecified	UNIVERSITAT POLITÈCNICA DE CATALUNYA BARCELONATECH Escola d'Enginyeria de Barcelona Est

UNIVERSITAT POLITÈCNICA DE CATALUNYA
BARCELONATECH
Escola d'Enginyeria de Barcelona Est

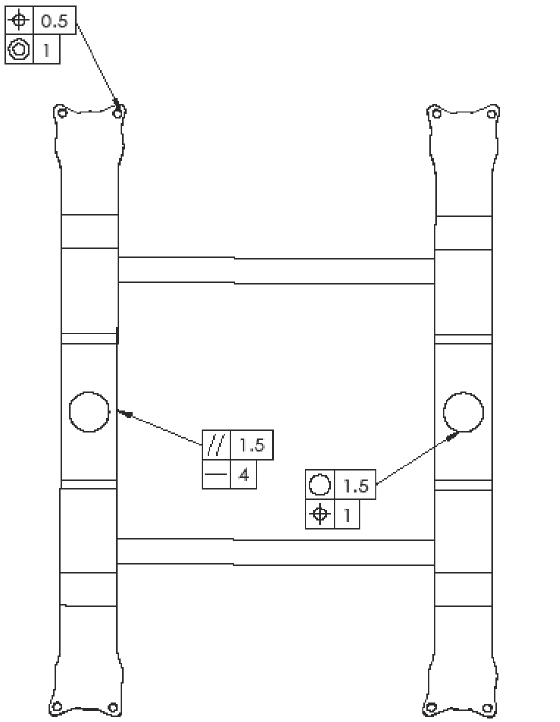


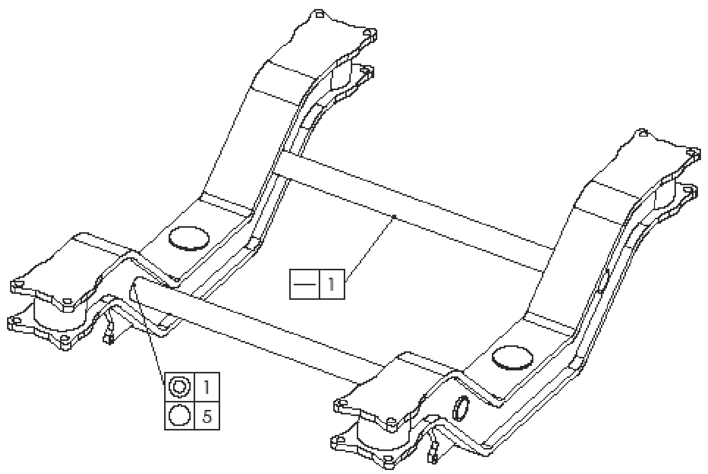
Date: 25/11/2020	Units: mm Unspecified tolerances: 0.5mm
Part: Bogie frame rod 1	Project titol: Tension study of a carbon fiber bogie frame
Drawing Numb.: 2 Quantity: 1	Draftman: Abelardo José Alfaro Checked by: Antonio José Sánchez Egea
Format: DIN A3 Scale: 1:20 Material: Structure Steel S235	UNIVERSITAT POLITÈCNICA DE CATALUNYA BARCELONATECH Escola d'Enginyeria de Barcelona Est

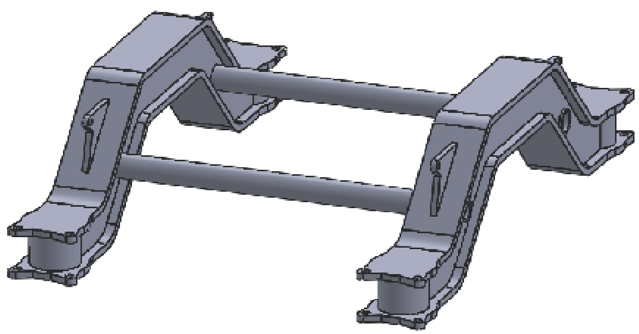


Date: 25/11/2020	Units: mm Unspecified tolerances: 0.5mm
Part: Bogie frame rod 2	Project titol: Tension study of a carbon fiber bogie frame
Drawing Numb.: 3 Quantity: 1	Draftman: Abelardo José Alfaro Checked by: Antonio José Sánchez Egea
Format: DIN A4 Scale: 1:20 Material: Composite fiber	UNIVERSITAT POLITÈCNICA DE CATALUNYA BARCELONATECH Escola d'Enginyeria de Barcelona Est

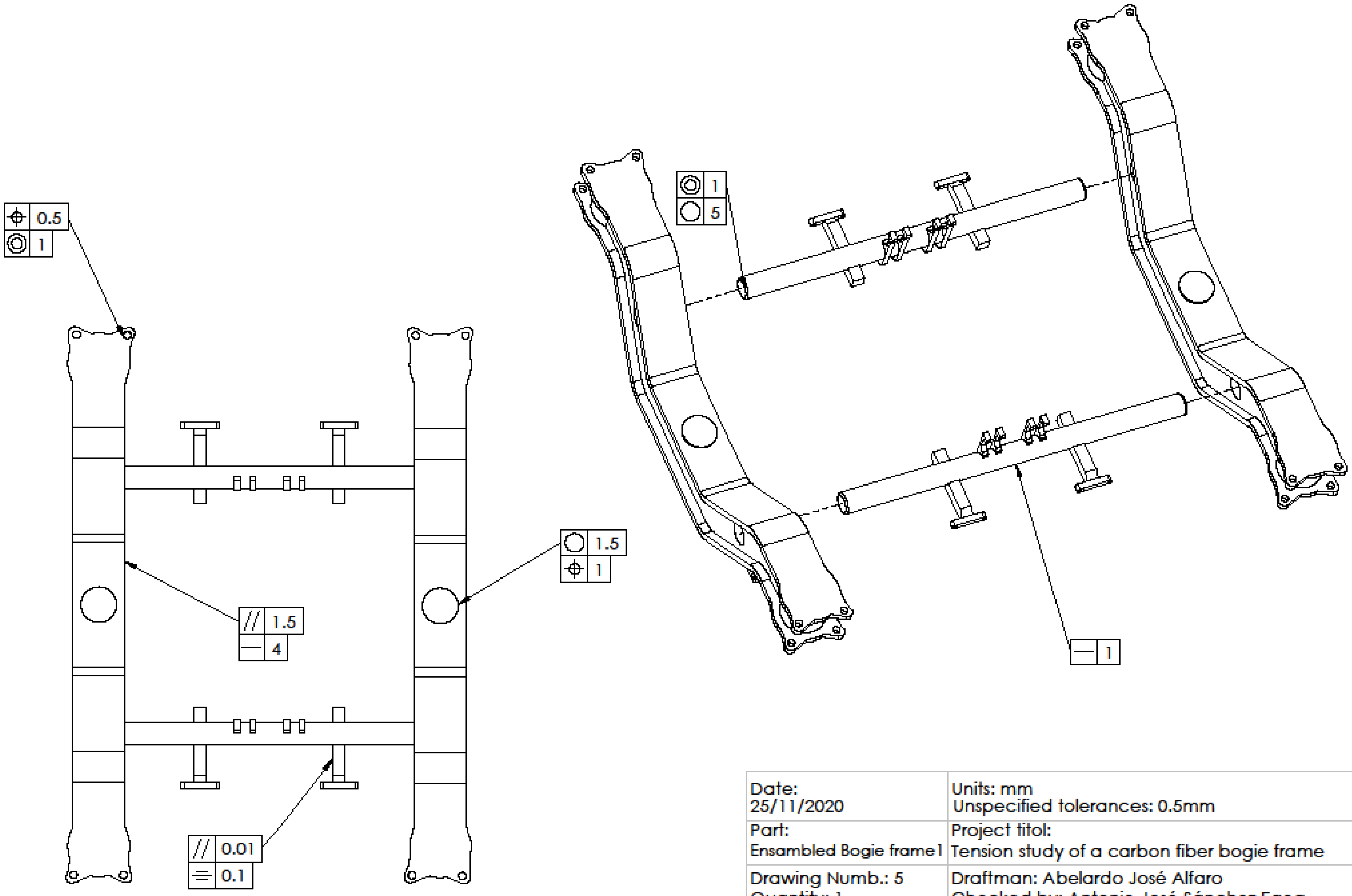
UNIVERSITAT POLITÈCNICA DE CATALUNYA
BARCELONATECH
Escola d'Enginyeria de Barcelona Est





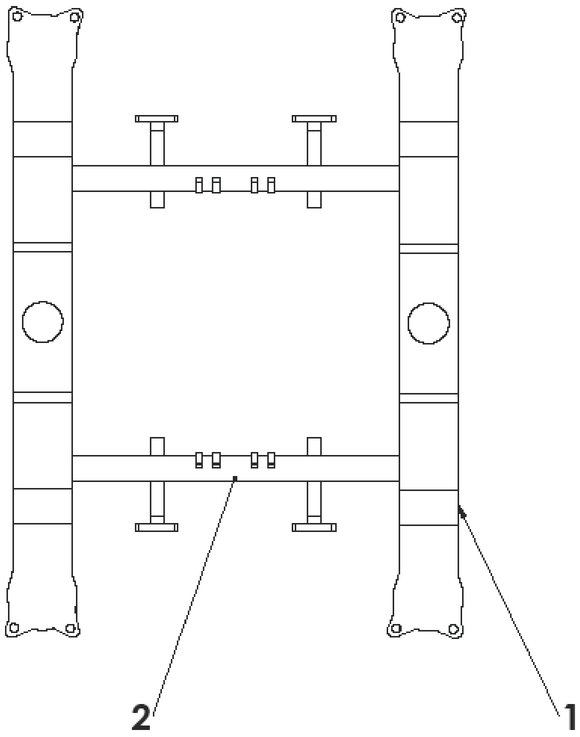


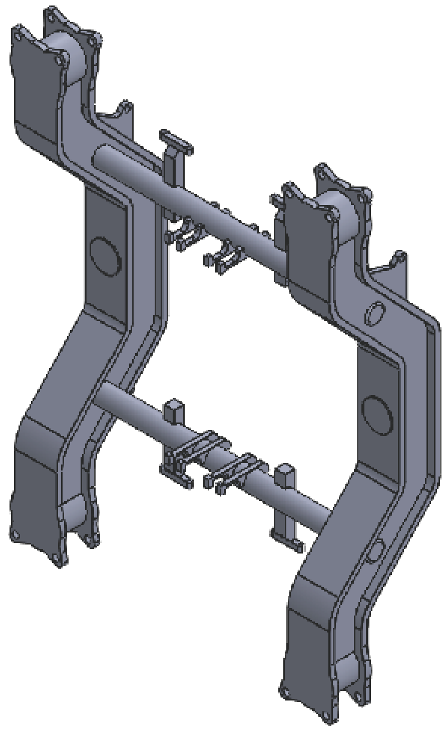
Date: 25/11/2020	Units: mm Unspecified tolerances: 0.5mm
Part: Ensambled bogie frame 2	Project titol: Tension study of a carbon fiber bogie frame
Drawing Numb.: 4 Quantity: 1	Draftman: Abelardo José Alfaro Checked by: Antonio José Sánchez Egea
Format: DIN A3 Scale: 1:20 Material: Composite fiber	UNIVERSITAT POLITÈCNICA DE CATALUNYA BARCELONATECH Escola d'Enginyeria de Barcelona Est



Date: 25/11/2020	Units: mm Unspecified tolerances: 0.5mm
Part: Ensambled Bogie frame1	Project titol: Tension study of a carbon fiber bogie frame
Drawing Numb.: 5 Quantity: 1	Draftman: Abelardo José Alfaro Checked by: Antonio José Sánchez Egea
Format: DIN A3 Scale: 1:20 Material: Structure Steel S235	UNIVERSITAT POLITÈCNICA DE CATALUNYA BARCELONATECH Escola d'Enginyeria de Barcelona Est

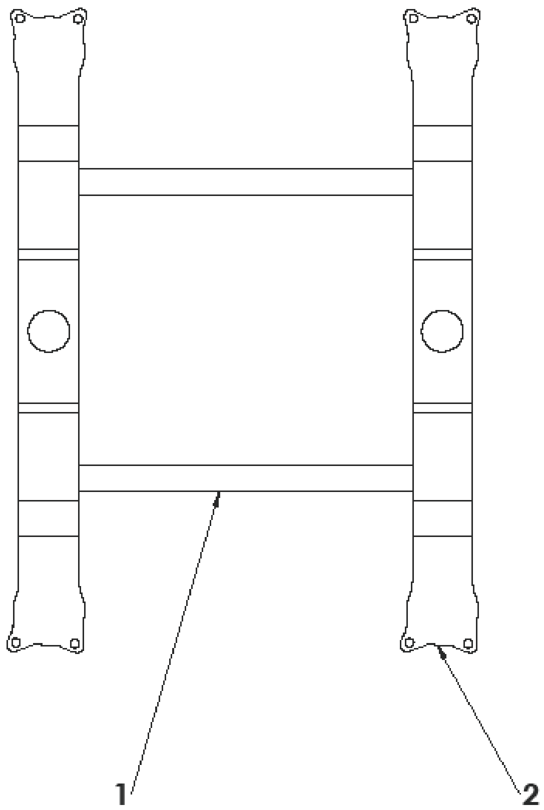
UNIVERSITAT POLITÈCNICA DE CATALUNYA
BARCELONATECH
Escola d'Enginyeria de Barcelona Est

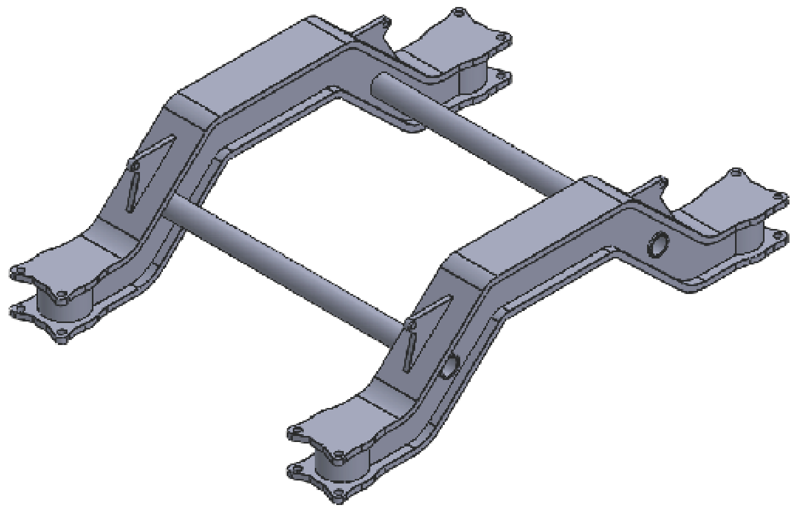




N.º DE ELEMENTO	N.º DE PIEZA	CANTIDAD
1	Frame solidwork	2
2	Frame-rod solidwork	2

Date: 25/11/2020	Units: mm Unspecified tolerances: 0.5mm
Part: Ensambled Bogie frame1	Project titol: Tension study of a carbon fiber bogie frame
Drawing Numb.: 6 Quantity: 1	Draftman: Abelardo José Alfaro Checked by: Antonio José Sánchez Egea
Format: DIN A3 Scale: 1:20 Material: Structure Steel S235	UNIVERSITAT POLITÈCNICA DE CATALUNYA BARCELONATECH Escola d'Enginyeria de Barcelona Est





N.º DE ELEMENTO	N.º DE PIEZA	CANTIDAD
1	Frame-rod solidwork - plate	2
2	Frame solidwork	2

Date: 25/11/2020	Units: mm Unspecified tolerances: 0.5mm
Part: Ensambled bogie frame 2	Project titol: Tension study of a carbon fiber bogie frame
Drawing Numb.: 7 Quantity: 1	Draftman: Abelardo José Alfaro Checked by: Antonio José Sánchez Egea
Format: DIN A3 Scale: 1:20 Material: Composite fiber	UNIVERSITAT POLITÈCNICA DE CATALUNYA BARCELONATECH Escola d'Enginyeria de Barcelona Est

To finalize appendix II, it is convenient to have a table that explains the geometrical tolerances that were present in the above technical drawings so that there is total comprehension.

Table 15. list of geometrical tolerances found in the technical drawing above.

Geometric characteristic	Symbol
Straightness	⏤
Symmetry	⌯
Parallelism	//
Perpendicularity	⊥
Coaxiality	◎
Positional	⌖
Circularity	○
Cylindricity	⌭

d. Appendix IIII - Technical data sheet of the materials in the study:

In this appendix IIII, we will go through the different technical data sheet of all the materials that took part in the study. All the tables that are present in this segment are taken from ANSYS report preview and are the properties that were used in the study of the bogie frame.

Structure Steel S235:

Staring of with the Structure Steel S235 which is a variation of structure steel of low carbon alloy. Ahead is the table describing in weight percentage the different elements that constitutes it.

Table 16. List of the materials that make up Structure steel S235.

Element	Weight %
Mn	1.4%
Cu	0.55%
C	0.17%
P	0.04%
S	0.035%
N	0.01%

The following table displays the mechanical and thermal properties that the Structure steel S235 has, it is important to have a full understanding of the mechanical behavior represented in the study.

Table 17. Technical data sheet of the Structure steel S235

Property	**Value**	**Units**
Density	7850	$kg \cdot m^{-3}$
Isotropic Secant Coefficient of Thermal Expansion	1.2e-005	$C^{\circ -1}$
Specific Heat Constant Pressure	434	$J\ kg^{-1} \cdot C^{\circ -1}$
Isotropic Thermal Conductivity	60.5	$W \cdot m^{-1} \cdot C^{\circ -1}$
Isotropic Resistivity	1.7e-007	$Ohm \cdot m$
Compressive Yield Strength	2.35e+008	Pa
Tensile Yield Strength	2.35e+008	Pa
Tensile Ultimate Strength	4.6e+008	Pa
Young's Modulus	2.35e+011	Pa
Poisson's Ratio	0.28	-
Shear Modulus	7.6923e+010	Pa
Relative Permeability	10000	-

Zero-Thermal-Strain Reference Temperature C	20	C°

Finally, the S-N curve for the structure steel S235 table will be displayed to give meaning to the life expectancy for the structure steel S235 bogie frame model.

Table 18. S-N curve for the Structure steel S235.

Alternating Stress Pa	Cycles
3.999e+009	10
2.827e+009	20
1.896e+009	50
1.413e+009	100
1.069e+009	200
4.41e+008	2000
2.62e+008	10000
2.14e+008	20000
1.38e+008	1.e+005
1.14e+008	2.e+005
8.62e+007	1.e+006

Woven Epoxy Carbon Prepreg:

The next material present in the study is the woven epoxy carbon prepreg being part of the materials that makes up the composite bogie frame. we will follow the same layout than the that of the structure steel s235.

Starting with the mechanical and thermal technical data sheet available in the ANSYS software we have that:

Table 19. Mechanical and Thermal properties of the Woven Epoxy Carbon Prepreg.

Property	Value	Units
Density	1420	$kg \cdot m^{-3}$
Isotropic Secant Coefficient of Thermal Expansion X	2.2e-006	$C^{\circ -1}$
Isotropic Secant Coefficient of Thermal Expansion Y	2.2e-006	$C^{\circ -1}$
Isotropic Secant Coefficient of Thermal Expansion Z	1.0e-005	$C^{\circ -1}$
Young's Modulus X Direction	6.134e+010	Pa
Young's Modulus Y Direction	6.134e+010	Pa
Young's Modulus Z Direction	6.9e+009	Pa
Poisson's Ratio XY	4.e-002	-
Poisson's Ratio YZ	0.3	-
Poisson's Ratio XZ	0.3	-
Shear Modulus XY	1.95e+010	Pa
Shear Modulus YZ	2.7e+009	Pa
Shear Modulus XZ	2.7e+009	Pa

Orthotropic Strain Limits - Tensile X direction	1.26e-002	-
Orthotropic Strain Limits - Tensile Y direction	1.26e-002	-
Orthotropic Strain Limits - Tensile Z direction	8.e-003	-
Orthotropic Strain Limits - Compressive X direction	-1.02e-002	-
Orthotropic Strain Limits - Compressive Y direction	-1.02e-002	-
Orthotropic Strain Limits - Compressive Z direction	-1.2e-002	-
Orthotropic Strain Limits - Shear XY	2.2e-002	-
Orthotropic Strain Limits - Shear YZ	1.9e-002	-
Orthotropic Strain Limits - Shear XZ	1.9e-002	-
Tensile X Direction	8.5e+008	Pa
Tensile Y Direction	8.5e+008	Pa
Tensile Z Direction	5.e+007	Pa
Compressive X Direction	-5.09e+008	Pa
Compressive Y Direction	-5.09e+008	Pa
Compressive Z Direction	-1.7e+008	Pa
Shear XY	1.25e+008	Pa
Shear YZ	6.5e+007	Pa
Shear XZ	6.5e+007	Pa
Zero-Thermal-Strain Reference Temperature C	20	C°

The fatigue properties were not taken into consideration for the making of the composite frame.

Recycled UD Epoxy Carbon Prepreg:

The following material is the recycled unidirectional epoxy carbon prepreg, it is also one of the carbon composite materials that makes up the composite bogie frame and as such, it is worth describing its mechanical and thermal properties of it.

The following table displays the thermal and mechanical property of the recycled unidirectional epoxy carbon prepreg. For the creation of the recycled unidirectional epoxy carbon prepreg material, the data accessible through ANSYS was modifies based on the results found in the study referenced in the memory, where in the study it is demonstrated that the recycled version lost 3.2% of the Young's Modulus and a 14% in the Tensile Strength. As such, the data available in the software ANSYS was modified reducing 14% the TS and 3.2% the YM to represent it in the simulation.

Table 20. List the mechanical and thermal properties for the Recycled UD epoxy carbon that was used in the simulation of the composite frame.

Property	Value	Units
Density	1490	$kg \cdot m^{-3}$
Isotropic Secant Coefficient of Thermal Expansion X	-4.7e-007	$C^{\circ -1}$
Isotropic Secant Coefficient of Thermal Expansion Y	3.e-005	$C^{\circ -1}$
Isotropic Secant Coefficient of Thermal Expansion Z	3.e-005	$C^{\circ -1}$
Young's Modulus X Direction	1.171e+011	Pa

Young's Modulus Y Direction	8.3226e+009	Pa
Young's Modulus Z Direction	8.3226e+009	Pa
Poisson's Ratio XY	0.27	-
Poisson's Ratio YZ	0.4	-
Poisson's Ratio XZ	0.27	-
Shear Modulus XY	4.7e+009	Pa
Shear Modulus YZ	3.1e+009	Pa
Shear Modulus XZ	4.7e+009	Pa
Orthotropic Strain Limits - Tensile X direction	1.67e-002	-
Orthotropic Strain Limits - Tensile Y direction	3.2e-003	-
Orthotropic Strain Limits - Tensile Z direction	3.2e-003	-
Orthotropic Strain Limits - Compressive X direction	-1.08e-002	-
Orthotropic Strain Limits - Compressive Y direction	-1.92e-002	-
Orthotropic Strain Limits - Compressive Z direction	-1.92e-002	-
Orthotropic Strain Limits - Shear XY	1.2e-002	-
Orthotropic Strain Limits - Shear YZ	1.1e-002	-
Orthotropic Strain Limits - Shear XZ	1.2e-002	-
Tensile X Direction	1.9128e+009	Pa
Tensile Y Direction	2.49e+007	Pa
Tensile Z Direction	2.49e+007	Pa
Compressive X Direction	-1.082e+009	Pa
Compressive Y Direction	-1e+008	Pa
Compressive Z Direction	-1e+008	Pa
Shear XY	6.e+007	Pa
Shear YZ	3.2e+007	Pa
Shear XZ	6.e+007	Pa
Zero-Thermal-Strain Reference Temperature	20	C°

Like before, the fatigue properties were not taken into consideration for the making of the composite frame.

Aluminum Honeycomb:

The next and last material present in the composite bogie frame, is the Aluminum honeycomb. Just like in the previous sections of this appendix, the mechanical properties of the aluminum honeycomb will be displayed.

The following table of properties is a list of mechanical properties that were used for the honeycomb and therefor to help construct the composite bogie frame.

Table 21. List the mechanical properties for the Aluminum honeycomb that was used in the simulation of the composite frame.

Property	Value	Units
Density	80	$Kg \cdot m^{-3}$
Young's Modulus X Direction	1.e+006	Pa
Young's Modulus Y Direction	1.e+006	Pa
Young's Modulus Z Direction	2.55e+008	Pa
Poisson's Ratio XY	0.49	-
Poisson's Ratio YZ	1.e-003	-
Poisson's Ratio XZ	1.e-003	-
Shear Modulus XY	1	Pa
Shear Modulus YZ	3.7e+007	Pa
Shear Modulus XZ	7.e+007	Pa
Orthotropic Strain Limits - Tensile X Direction	0	-
Orthotropic Strain Limits - Tensile Y Direction	0	-
Orthotropic Strain Limits - Tensile Z Direction	5.31e+006	-
Orthotropic Strain Limits - Compressive X Direction	0	-
Orthotropic Strain Limits - Compressive Y Direction	0	-
Orthotropic Strain Limits - Compressive Z Direction	-5.31e+006	-
Orthotropic Strain Limits - Shear XY	0	-
Orthotropic Strain Limits - Shear YZ	1.21e+006	-
Orthotropic Strain Limits - Shear XZ	2.24e+006	-