

Autodesk Nastran Analysis Report

Date: 06/19/18
Author: Autodesk Customer
Subject: Analysis
Prepared For: Autodesk Customer
Software Used: Autodesk Nastran Version 12.1.0.925

Tables of Contents

[1. Summary](#)

[2. Assumptions](#)

[3. Model Definition](#)

[3.1 Group Definition](#)

[Table 3.1.1 Group Definition](#)

[Table 3.1.2 Part Mass Properties](#)

[3.2 Contact Definition](#)

[3.3 Material Properties](#)

[Table 3.3.1 Isotropic Material Definition](#)

[Table 3.3.2 Anisotropic Shell Element Material Definition](#)

[Table 3.3.3 Anisotropic Solid Element Material Definition](#)

[Table 3.3.4 Orthotropic Shell Element Material Definition](#)

[Table 3.3.5 Orthotropic Solid Element Material Definition](#)

[Table 3.3.6 Hyperelastic Element Material Definition](#)

[3.4 Mesh](#)

[Table 3.4.1 Element Initial Distortion Summary](#)

[4. Environment](#)

[4.1 Structural Loading](#)

[Table 4.1.1 Applied Load Vector Resultant](#)

[4.2 Structural Support](#)

[Table 4.2.1 Reaction Load Vector Resultant](#)

[5. Solution](#)

[Table 5.1.1 Displacement Summary](#)

[Table 5.1.2 Peak Displacement Component Summary](#)

[Table 5.1.3 Stress Results Summary](#)

[Table 5.1.4 Solution Error Measure and the Relative Stress Error Summary](#)

[6. Conclusion](#)

[7. Glossary](#)

1. Summary

The report documents design and analysis using Autodesk Nastran engineering simulation software. A linear static analysis was performed using the finite element model shown in the figure below. The model is divided into 5 property group(s). The units system is mm-N-s. The model consists of a total of 133902 nodes and 82552 elements.

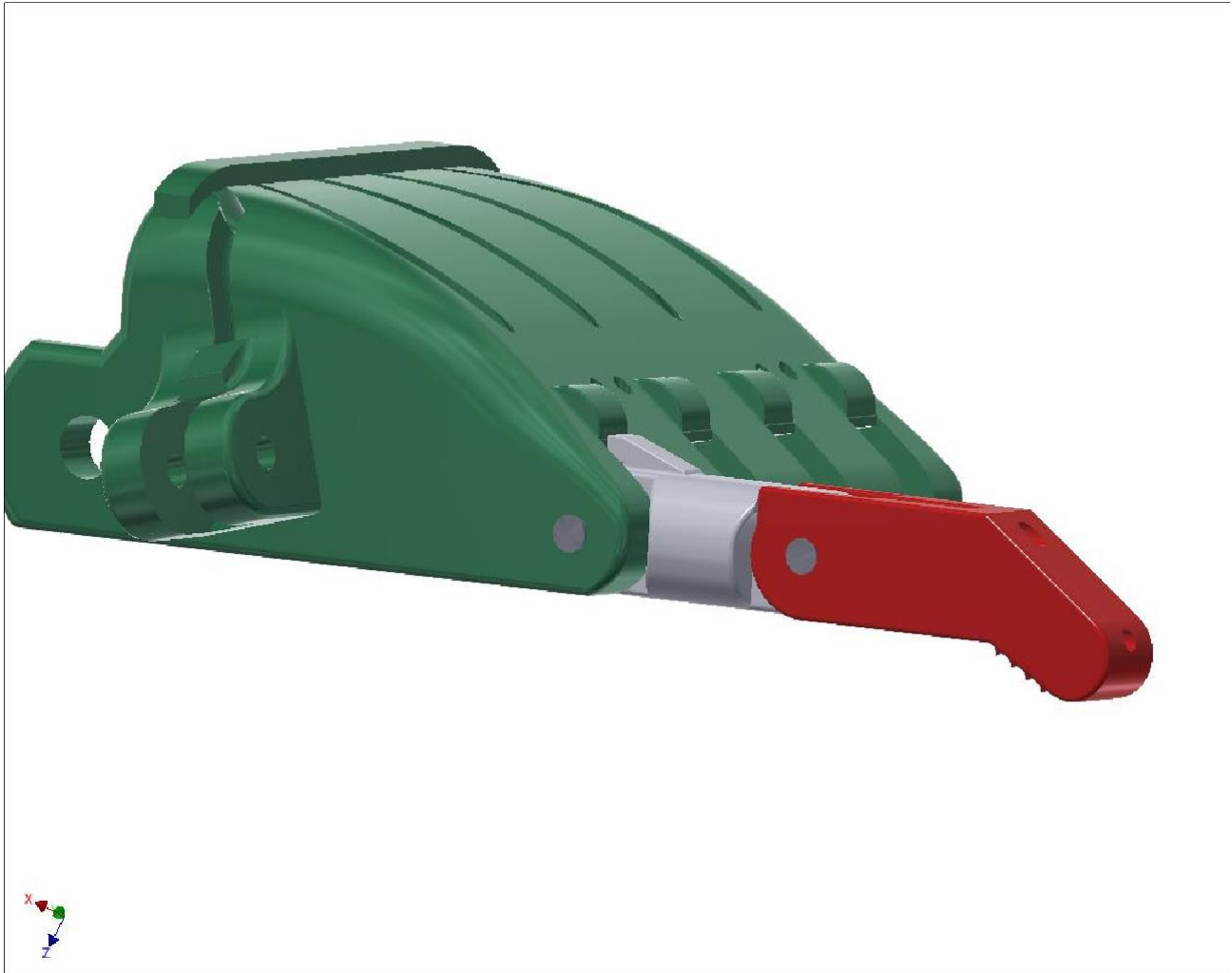


Figure 1 - Finite Element Model

2. Assumptions

1. Displacements are small.
2. [Follower forces](#) are ignored.

3. Model Definition

3.1 Group Definition

- The model is divided into 5 property group(s). Details for each group are given in Table 3.1.1.
1. The [bounding box](#) for all positioned bodies in the model measures 85,9 by 90,79 by 8,65mm along the basic coordinate system x, y and z axes, respectively.
 2. The total mass of the model is 5,23E-05 t.
 3. The model center of mass is located at (, ,) mm.

Table 3.1.1 Group Definition

Property Group	Material	Bounding Box (mm)	Mass (t)	Volume (mm³)	Nodes	Elements
SOLID 2	MAT 2	85,9, 90,79, 50,63	4,485E-05	3,588E+04	109774	68805
SOLID 3	MAT 2	25,32, 27,5, 18,71	2,618E-06	2095,0	9127	5495
SOLID 4	MAT 2	34,7, 34,26, 19,03	3,288E-06	2631,0	12153	7289
SOLID 5	MAT 3	8,232, 8,276, 5,567	4,487E-07	56,08	557	359
SOLID 6	MAT 3	16,39, 16,61, 8,65	1,091E-06	136,3	781	604

Table 3.1.2 Part Mass Properties

No Data

3.2 Contact Definition

- The model contains 9 contact region(s).
- Adaptive stiffness scaling is enabled.

Table 3.2.1 Contact Definition

Name	Type	Contact Surface	Normal Stiffness	Penetration
Contact Region 33	Offset Welded Contact	Surface 35, Surface 34	Stiffness Controlled	-
Contact Region 36	Offset Welded Contact	Surface 38, Surface 37	Stiffness Controlled	-
Contact Region 39	Offset Welded Contact	Surface 41, Surface 40	Stiffness Controlled	-
Contact Region 42	Offset Welded Contact	Surface 44, Surface 43	Stiffness Controlled	-
Contact Region 45	Offset Welded Contact	Surface 47, Surface 46	Stiffness Controlled	-
Contact Region 48	General Contact	Surface 50, Surface 49	Stiffness Controlled	Unsymmetric
Contact Region 51	General Contact	Surface 53, Surface 52	Stiffness Controlled	Unsymmetric
Contact Region 54	General Contact	Surface 56, Surface 55	Stiffness Controlled	Unsymmetric
Contact Region 57	Rough Contact	Surface 59, Surface 58	Stiffness Controlled	Symmetric

3.3 Material Properties

3.3.1 Isotropic Material Definition

Material ID	E	G	NU	RHO	ALPHA	T-REF
2	2346,0	862,7	0,36	1,25E-09	0,0	0,0
3	1,93E+05	7,423E+04	0,3	8,E-09	1,E-05	0,0

3.3.2 Anisotropic Shell Element Material Definition

No Data

3.3.3 Anisotropic Solid Element Material Definition

No Data

3.3.4 Orthotropic Shell Element Material Definition

No Data

3.3.5 Orthotropic Solid Element Material Definition

No Data

3.3.6 Hyperelastic Element Material Definition

No Data

3.4 Mesh

The finite element mesh is shown in the figure below. The model consists of a total of 133902 nodes and 82552 elements.

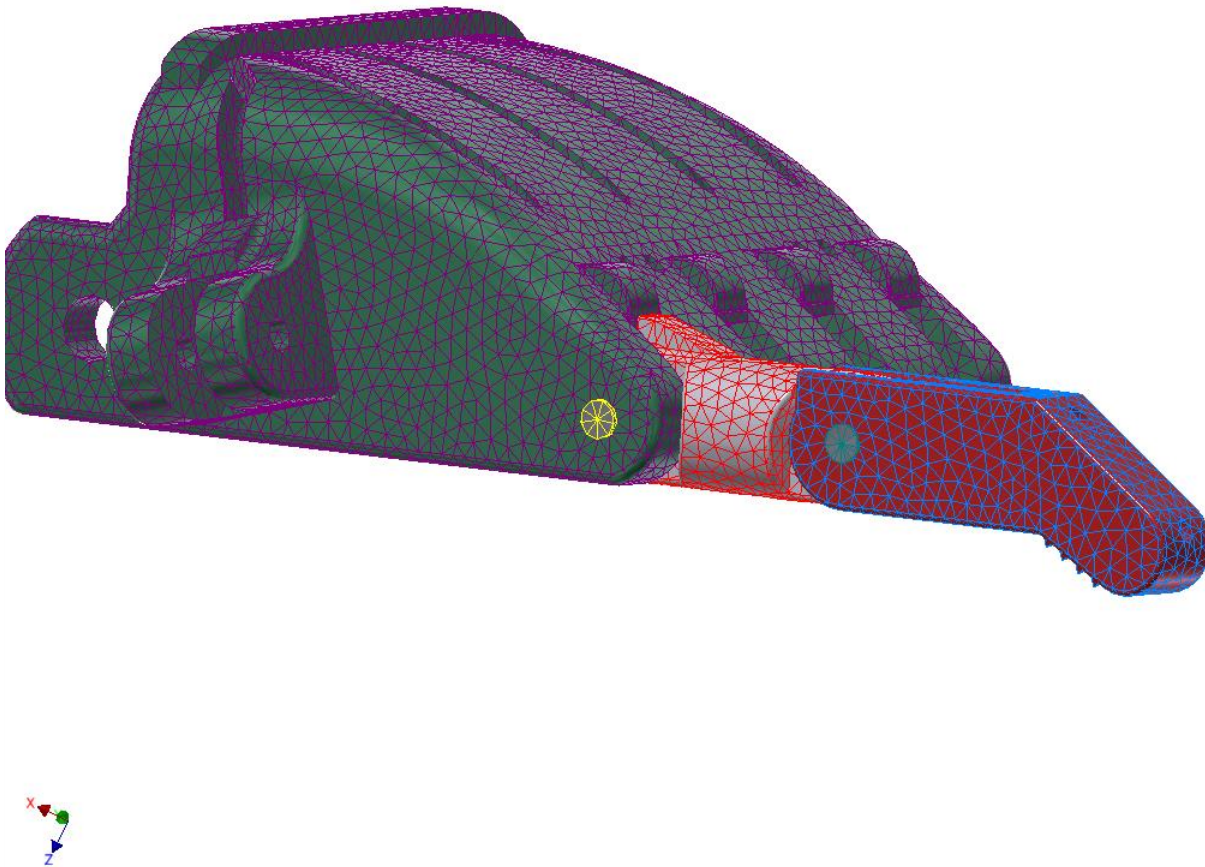


Figure 2 - Finite Element Mesh

4. Environment

4.1 Structural Loading

The finite element environments are shown in the figures below. Applied structural loading is summarized in Table 4.1.1. Applied load vector resultants are defined in the basic coordinate system. Moments are summed about location (0,0,0,0,0,0).

Table 4.1.1 Applied Load Vector Resultant

Subcase	Resultant Force(N)			Resultant Moment(N mm)		
	XT	YT	ZT	XR	YR	ZR
SUBCASE 1	0,2774	-3,301	-9,435	-42,14	454,6	-160,3

4.2 Structural Support

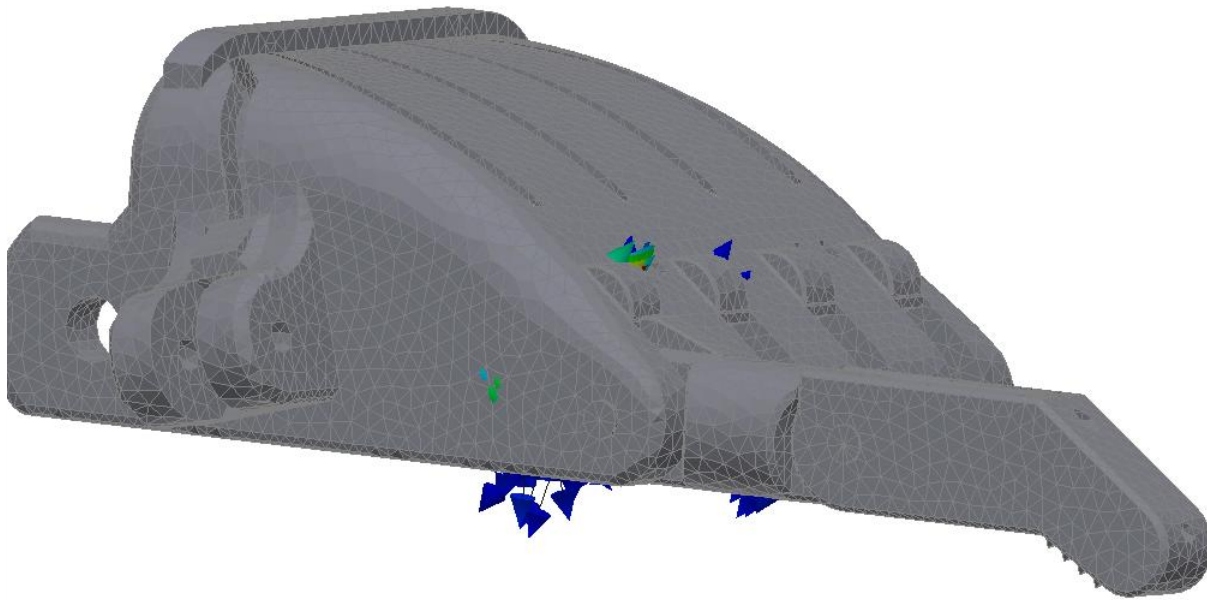
Reaction loads are summarized in Table 4.2.1. Reaction load vector resultants are defined in the basic coordinate system. Moments are summed about location (0,0,0,0,0,0).

Table 4.2.1 Reaction Load Vector Resultant

Subcase	Resultant Force (N)			Resultant Moment(N mm)		
	XT	YT	ZT	XR	YR	ZR
SUBCASE 1	-4,988E-02	4,945E-02	-9,794E-02	-1,745	10,51	5,749



Figure 3 - Applied Load



VECTOR PLOT: SPC FORCE; COMPONENTS: MAGNITUDE
OUTPUT SET: SUBCASE 1

Figure 4 - Reaction Load

5. Solution

The solution to the Environment defined in Section 4 applied to the Model defined in Section 3 is given below. The program selected the PCGLSS linear solver. Total solution time was 236.0 seconds. The largest [solution error measure](#) was 5,455E-09 for SUBCASE 1. The largest solid element [relative stress error](#) was 0,1125 for SUBCASE 1. The results are summarized in the table(s) and figure(s) below.

Table 5.1.1 Displacement Summary

Subcase	Minimum Displacement (mm)	Property Group	Maximum Displacement (mm)	Property Group
Subcase 1	0,109	Pasador:1	0,1428	Pasador:1
Subcase 1	1,983E-03	Pasador palma:1	2,504E-03	Pasador palma:1
Subcase 1	0,0	Palma:1	4,158E-04	Palma:1
Subcase 1	6,575E-02	Falange2ª-3ª:1	0,6059	Falange2ª-3ª:1
Subcase 1	1,404E-05	Falange1ª:1	0,1656	Falange1ª:1
Subcase 1	0,0		0,6059	

Table 5.1.2 Peak Displacement Component Summary

Subcase	Displacement Components (mm)			Rotation Components (mm)		
	XT	YT	ZT	XR	YR	ZR
SUBCASE 1	0,19	9,905E-02	0,5827	3,841E-02	3,983E-02	3,57E-02

Table 5.1.3 Stress Results Summary

Subcase	Minimum Principal Stress (MPa)	Property Group	Maximum Principal Stress (MPa)	Property Group	Maximum Von Mises Stress (MPa)	Property Group
Subcase 1	-12,94	Pasador:1	12,41	Pasador:1	13,68	Pasador:1
Subcase 1	-0,2862	Pasador palma:1	0,2043	Pasador palma:1	0,247	Pasador palma:1
Subcase 1	-0,4492	Palma:1	0,4093	Palma:1	0,7619	Palma:1
Subcase 1	-20,46	Falange2ª-3ª:1	10,1	Falange2ª-3ª:1	16,56	Falange2ª-3ª:1
Subcase 1	-22,41	Falange1ª:1	9,02	Falange1ª:1	18,65	Falange1ª:1
Subcase 1	-22,41		12,41		18,65	

Table 5.1.4 [Solution Error Measure](#) and the [Relative Stress Error](#) Summary

Subcase	Solution Error Measure	Shell Element Relative Stress Error	Solid Element Relative Stress Error
SUBCASE 1	5,455E-09	n/a	0,1125

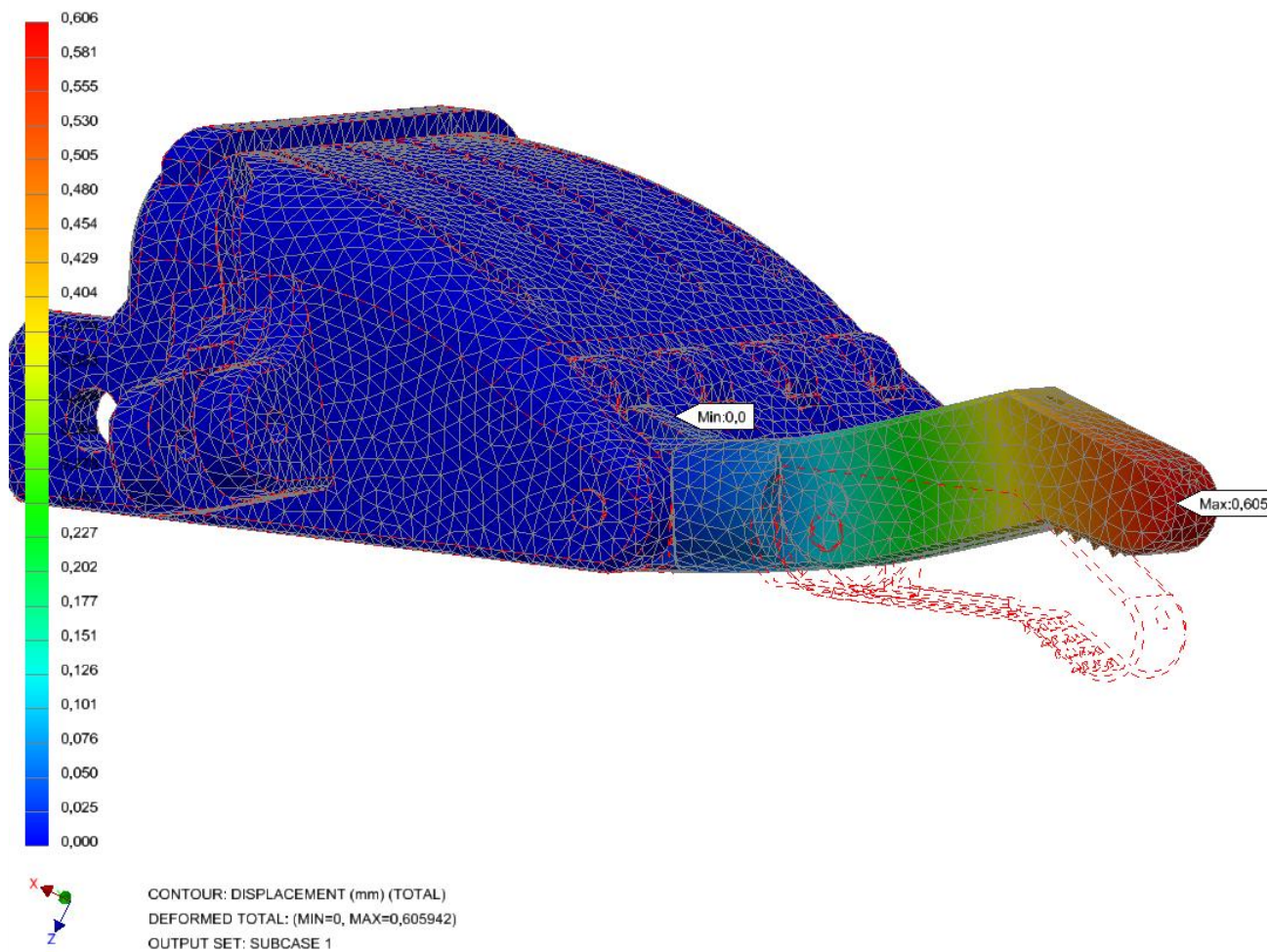


Figure 5 - OUTPUT SET: SUBCASE 1 -- DEFORMED TOTAL: (MIN=0, MAX=0,605942) -- CONTOUR: DISPLACEMENT (mm) (TOTAL)

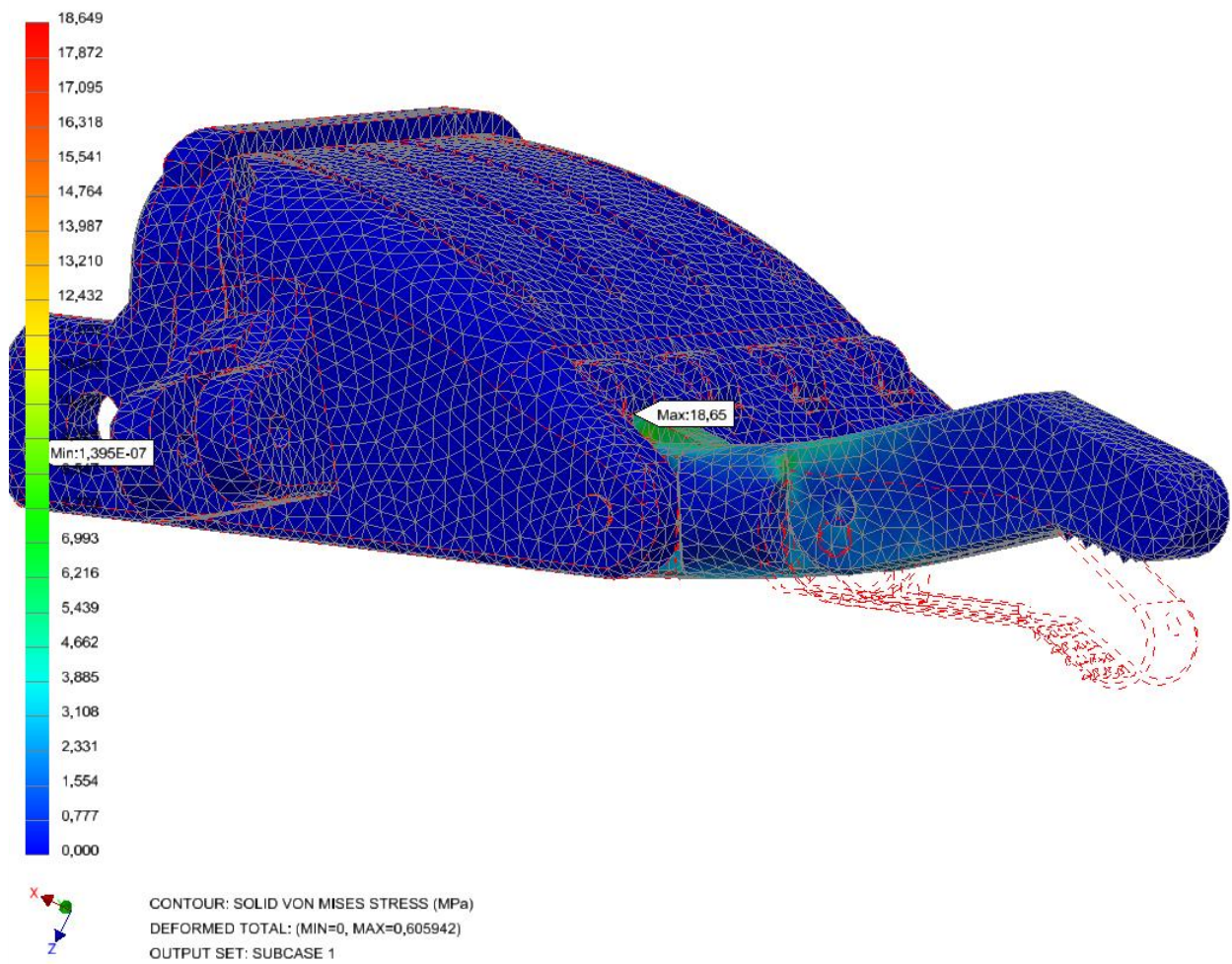


Figure 6 - OUTPUT SET: SUBCASE 1 -- DEFORMED TOTAL: (MIN=0, MAX=0,605942) -- CONTOUR: SOLID VON MISES STRESS (MPa)

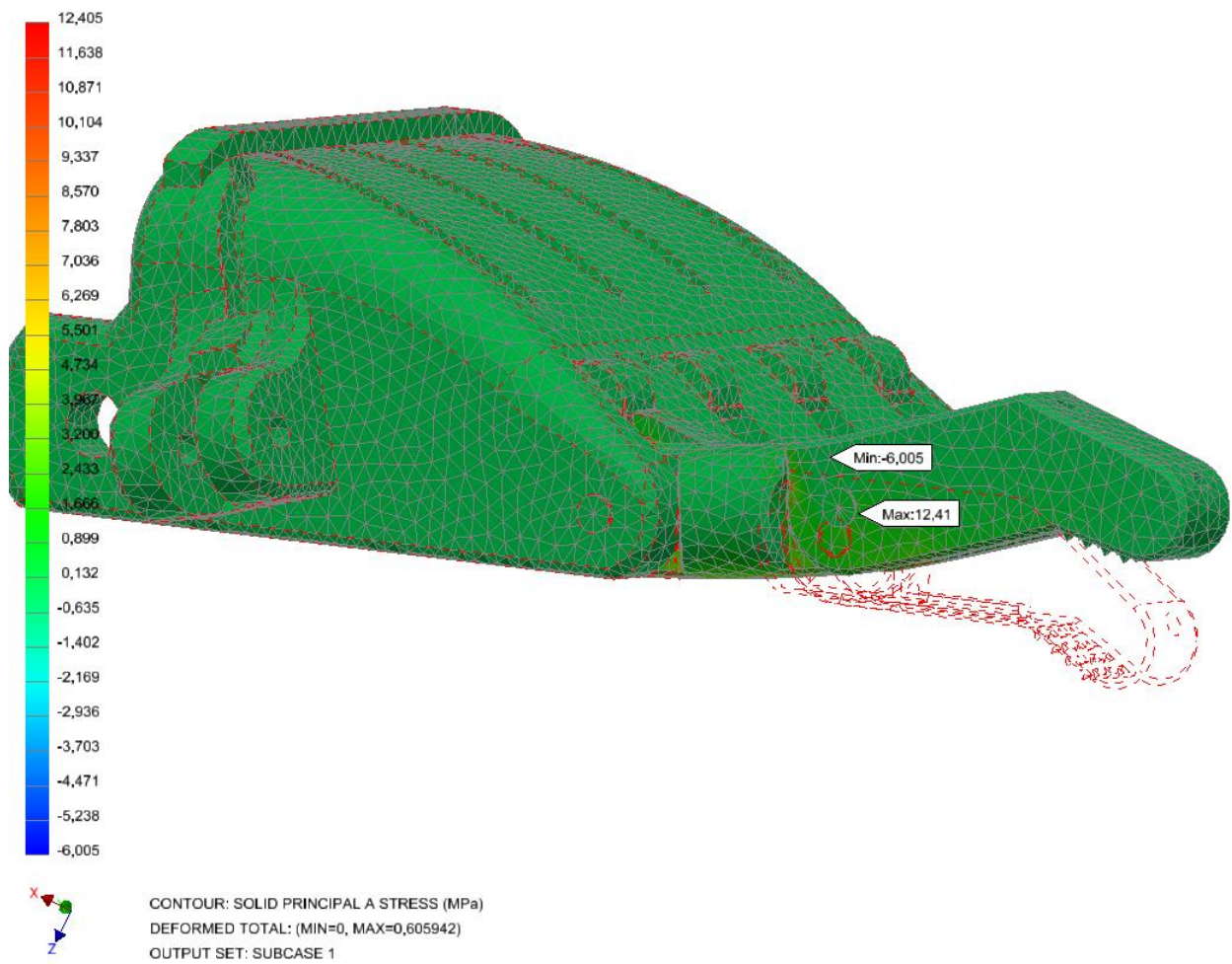


Figure 7 - OUTPUT SET: SUBCASE 1 -- DEFORMED TOTAL: (MIN=0, MAX=0,605942) -- CONTOUR: SOLID PRINCIPAL A STRESS (MPa)

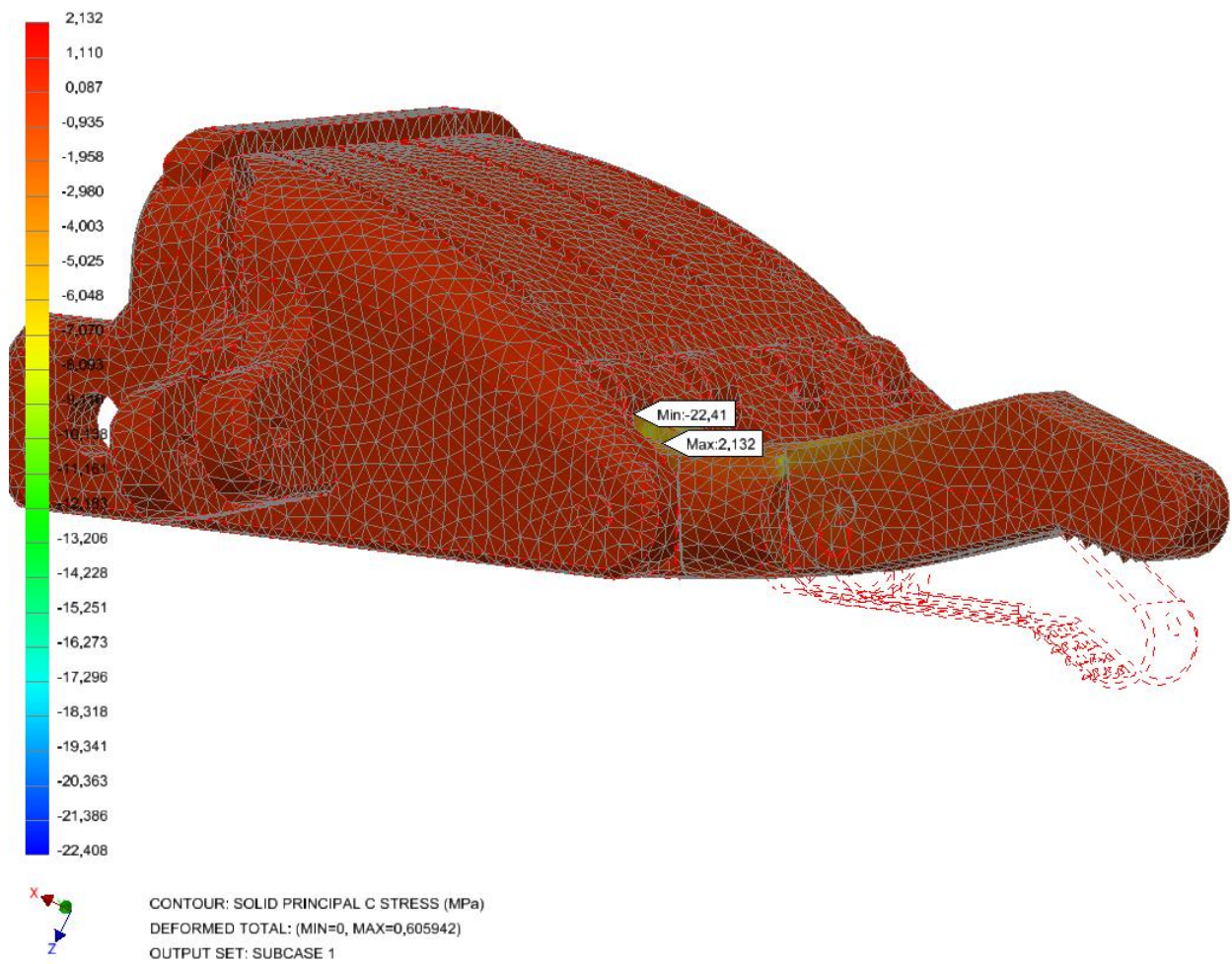


Figure 8 - OUTPUT SET: SUBCASE 1 -- DEFORMED TOTAL: (MIN=0, MAX=0,605942) -- CONTOUR: SOLID PRINCIPAL C STRESS (MPa)

6. Conclusion:

A linear static analysis was performed using the Autodesk Nastran Version 12.1.0.925 finite element solver on the o4wqwi8cd structure. The finite element model contained mainly Palma:1 elements and consisted of 407916 degrees of freedom.1 loading condition was analyzed.The maximum displacement was 0,1428 mm (load case Subcase 1)The maximum von Mises stress was 13,68 (load case Subcase 1).

7. Glossary:

Aspect Ratio

Ratio of an element's longest side to its adjacent side.

Bi-Directional Slide

Prevents contacting regions from separating or closing but permits sliding (zero coefficient of friction)

Bounding Box

A three-dimensional cube aligned to the global x,y and z axes that exactly contains a body or assembly.

Follower Force

Loads that follow the motion of the structure as it deforms.

General Contact

Models standard nonlinear surface contact with friction if specified.

Relative Stress Error

A measure of mesh convergence (values greater than 0.01 may indicate that further mesh refinement is required in areas with large stress gradients over a few elements).

Rough Contact

Nonlinear contact that allows separation and closure but does not permit sliding (infinite friction).

Skew Angle

The angle between the lines that join opposite midsides of a quadrilateral face.

Solution Error Measure

A measure of solution quality (values less than 1.0E-07 are generally considered acceptable).

Taper Ratio

The ratio of the areas on the two sides of a diagonal of a quadrilateral face.

Warping Angle

The extent to which a quadrilateral face deviates from being planar.

Welded Contact

Prevents contacting regions from sliding, separating, or closing.