Kernel Report Smooth 3D

Date: 11/24/24
Author: Fabio Pinheiro Cardoso
Subject: Analysis Report
Prepared For: Fabio Pinheiro Cardoso
Software Used: Autodesk Nastran Version 18.2.0.35

Tables of Contents

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 1 property group(s). The units system is m-N-s. The model consists of a total of 216776 nodes and 123300 elements.

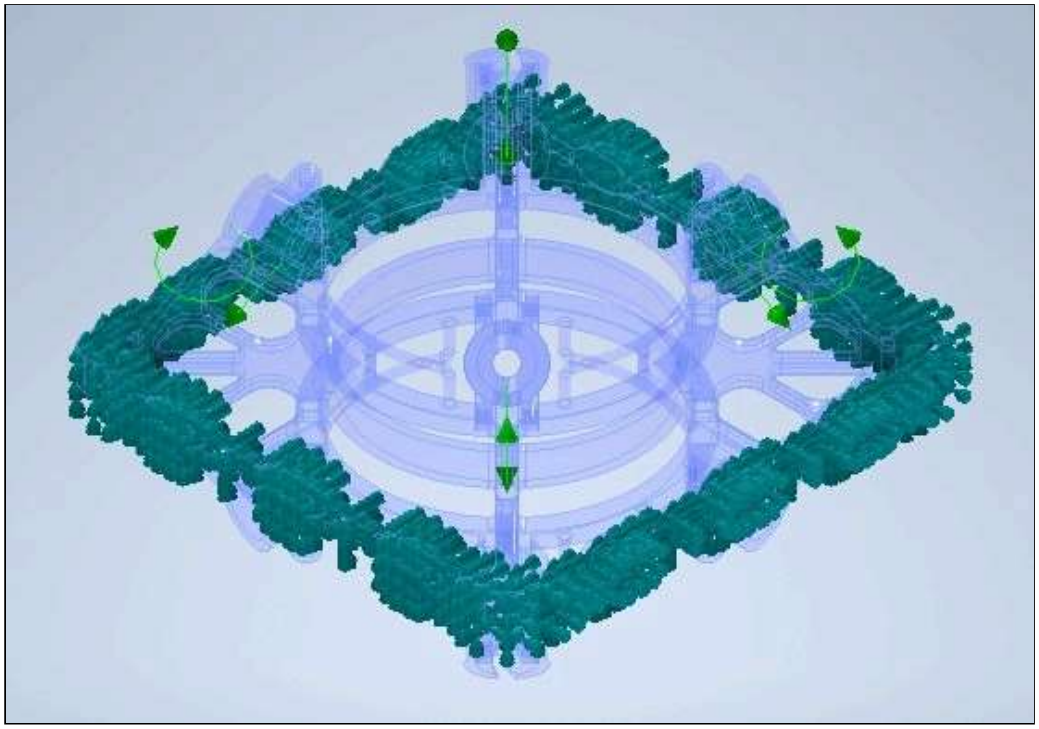

Figure 1 - Finite Element Model

2. Assumptions

1. Displacements are small.
2. Follower forces are ignored.

3. Model Definition

3.1 Group Definition

The model is divided into 1 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 0,1477 by 0,1043 by 0,1477m along the basic coordinate system x, y and z axes, respectively.
2. The total mass of the model is 0,2814 kg.
3. The model center of mass is located at (1,301E-02, 1,299E-02, -1,298E-02) m.

Table 3.1.1 Group Definition

Property Group	Material	Bounding Box (m)	Mass (kg)	Volume (m^3)	Nodes	Elements
SOLID 11	MAT 7	0,1477, 0,1043, 0,1477	0,2814	1,116E-04	216776	123300

Table 3.1.2 Part Mass Properties

Property Group	Material	Mass (kg)	Center of Mass (m)	Moments of Inertia (m)
SOLID 11	MAT 7	0,2814	1,301E-02, 1,299E-02, -1,298E-02	6,733E-04, 1,027E-03, 6,733E-04

3.2 Contact Definition

The model contains 0 contact region(s).
- Adaptive stiffness scaling is enabled.

3.3 Material Properties

3.3.1 Isotropic Material Definition

Material ID	E	G	NU	RHO	ALPHA	T-REF
7	1,566E+09	5,325E+08	0,47	2521,0	5,6E-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 216776 nodes and 123300 elements.

Table 3.4.1 Element Initial Distortion Summary

Property Group	Property Type	Aspect Ratio	Recommended Limit	Taper Ratio	Recommended Limit	Skew Angle	Recommended Limit	Warping Angle	Recommended Limit
SOLID 11	TET	15,38	100,0	0,0	0,0	175,1	80,0	0,0	0,0

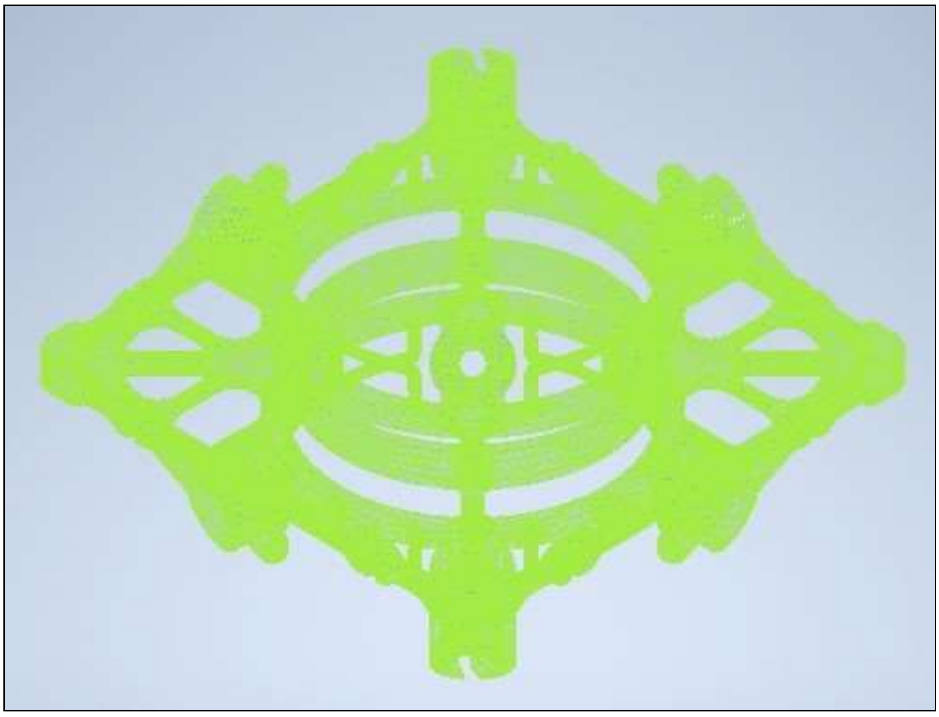

Figure 2 - Finite Element Mesh

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

	Resultant Force(N)			Resultant Moment(N m)		
Subcase	**XT**	**YT**	**ZT**	**XR**	**YR**	**ZR**
SUBCASE 6	0,1198	-48,93	2,493E-02	-0,6335	-6,345	-0,642

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

	Resultant Force (N)			Resultant Moment(N m)		
Subcase	**XT**	**YT**	**ZT**	**XR**	**YR**	**ZR**
SUBCASE 6	-0,1196	48,93	-2,489E-02	0,6335	6,345	0,642

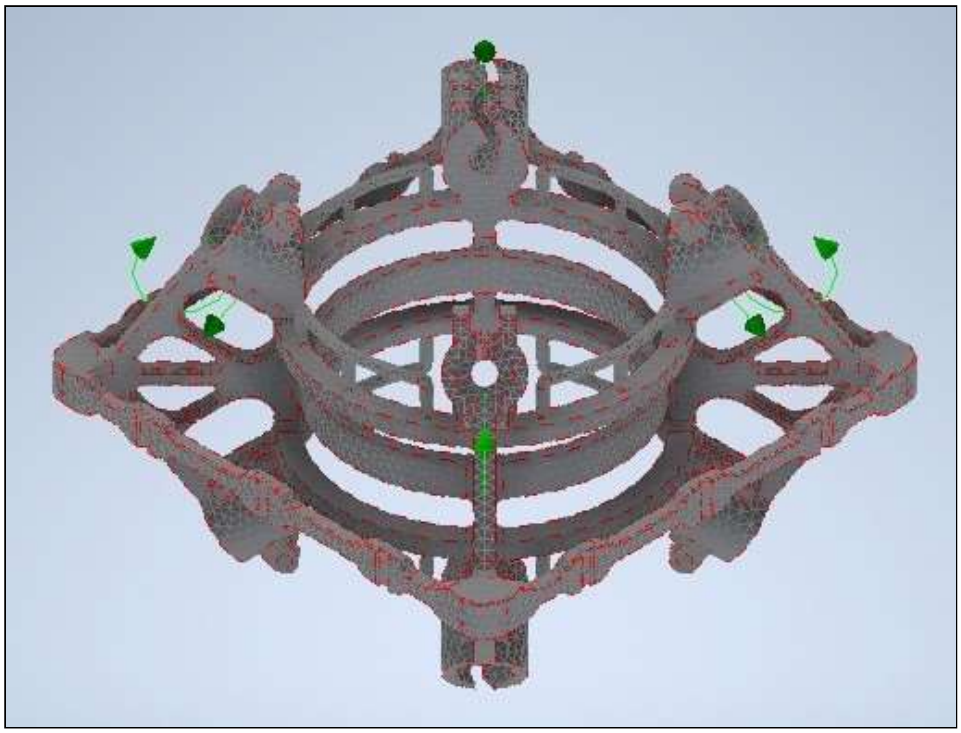

Figure 3 - Applied Load

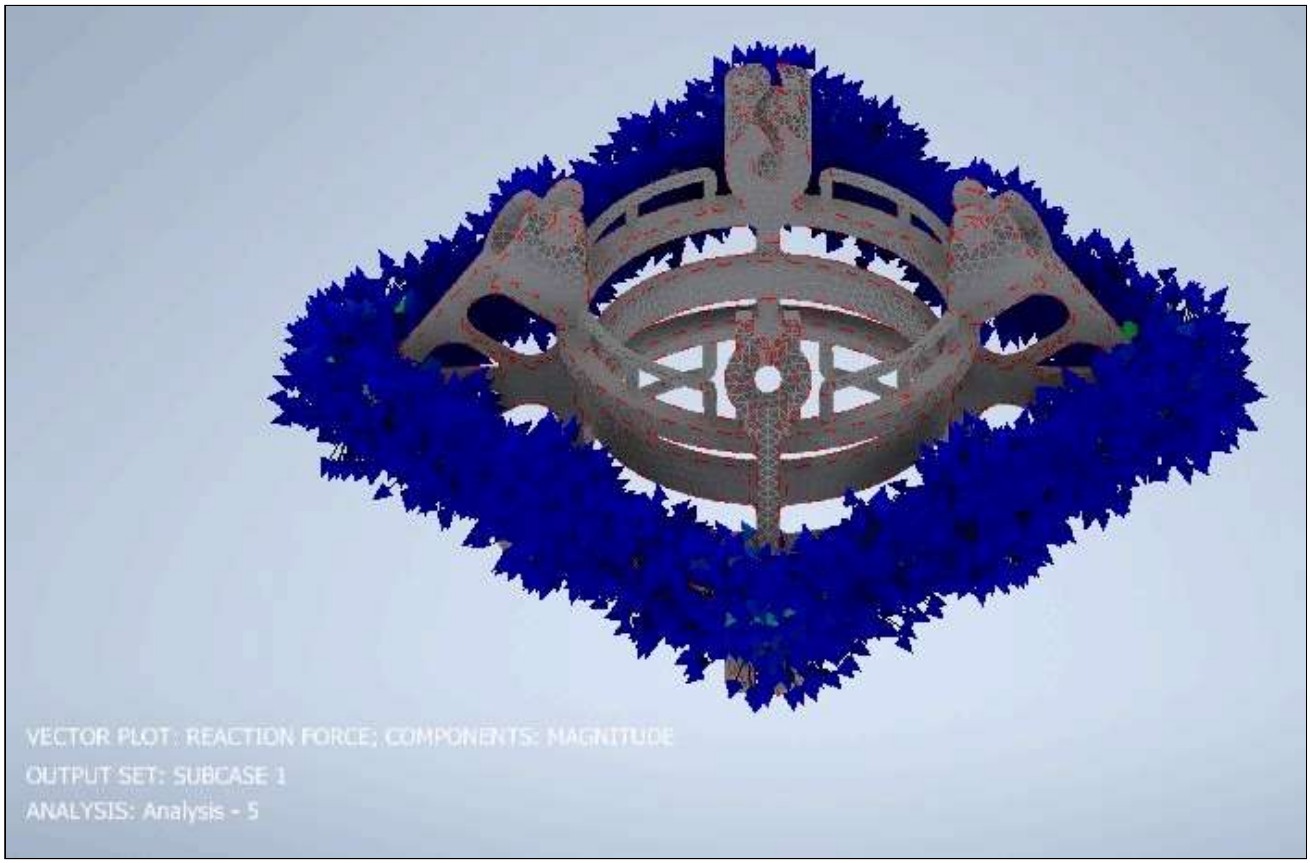

Figure 4 - Reaction Load

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 60.32 seconds.The largest solution error measure was 3,629E-09 for SUBCASE 6.The largest solid element relative stress error was 7,088E-02 for SUBCASE 6.The results are summarized in the table(s) and figure(s) below.

Table 5.1.1 Displacement Summary

Subcase	Minimum Displacement (m)	Property Group	Maxmium Displacement (m)	Property Group
Subcase 1	0,0		8,636E-03	

Table 5.1.2 Peak Displacement Component Summary

	Displacement Components (m)			Rotation Components (m)		
Subcase	XT	YT	ZT	XR	YR	ZR
SUBCASE 6	7,591E-03	3,858E-03	4,622E-03	0,0	0,0	0,0

Table 5.1.3 Stress Results Summary

Subcase	Minimum Principal Stress (Pa)	Property Group	Maximum Principal Stress (Pa)	Property Group	Maximum Von Mises Stress (Pa)	Property Group
Subcase 1	-1,041E+08		6,759E+07		8,488E+07	

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 6	3,629E-09	n/a	7,088E-02

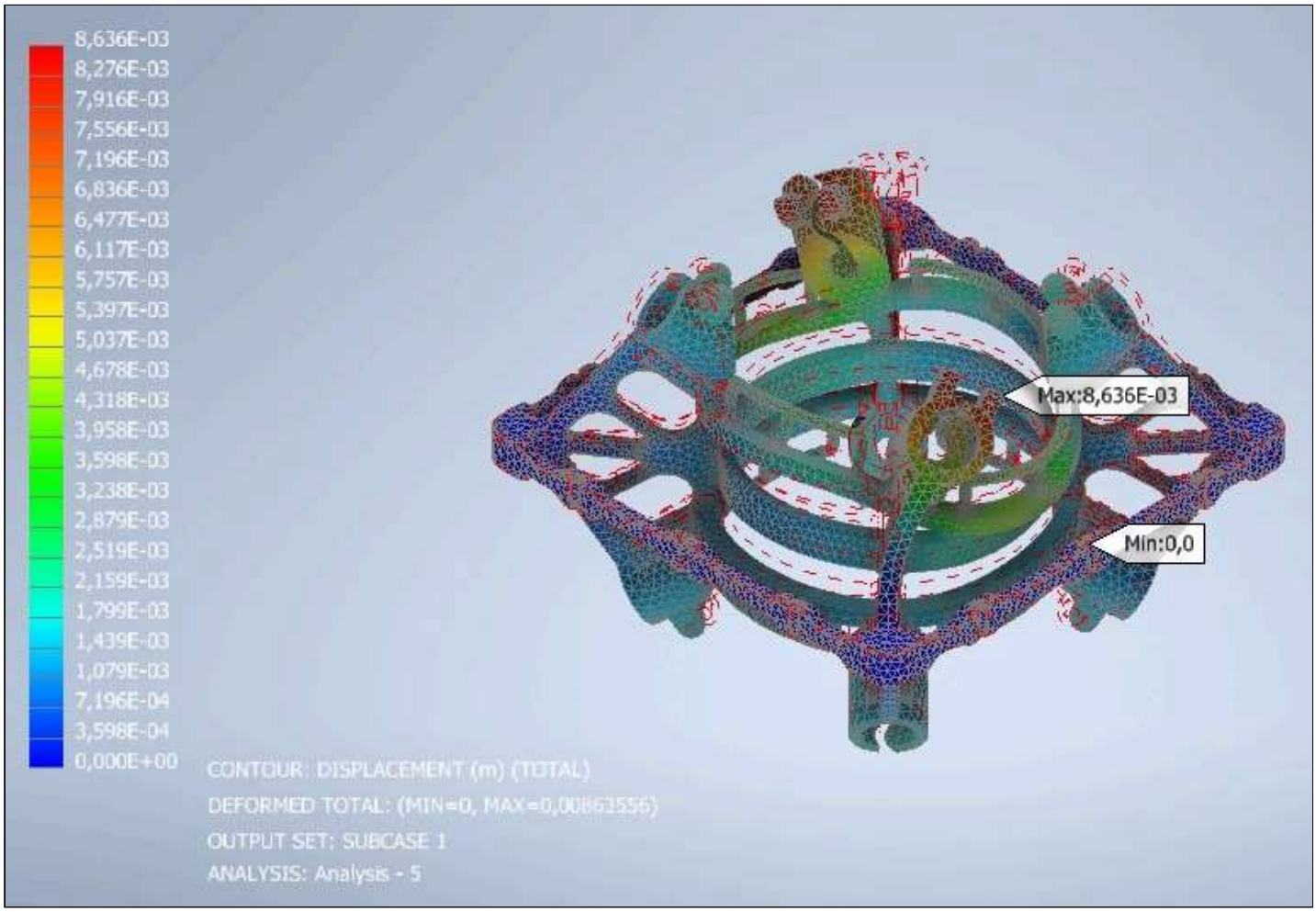

Figure 5 - OUTPUT SET: SUBCASE 1 -- DEFORMED TOTAL: (MIN=0, MAX=0,00863556) -- CONTOUR: DISPLACEMENT (m) (TOTAL)

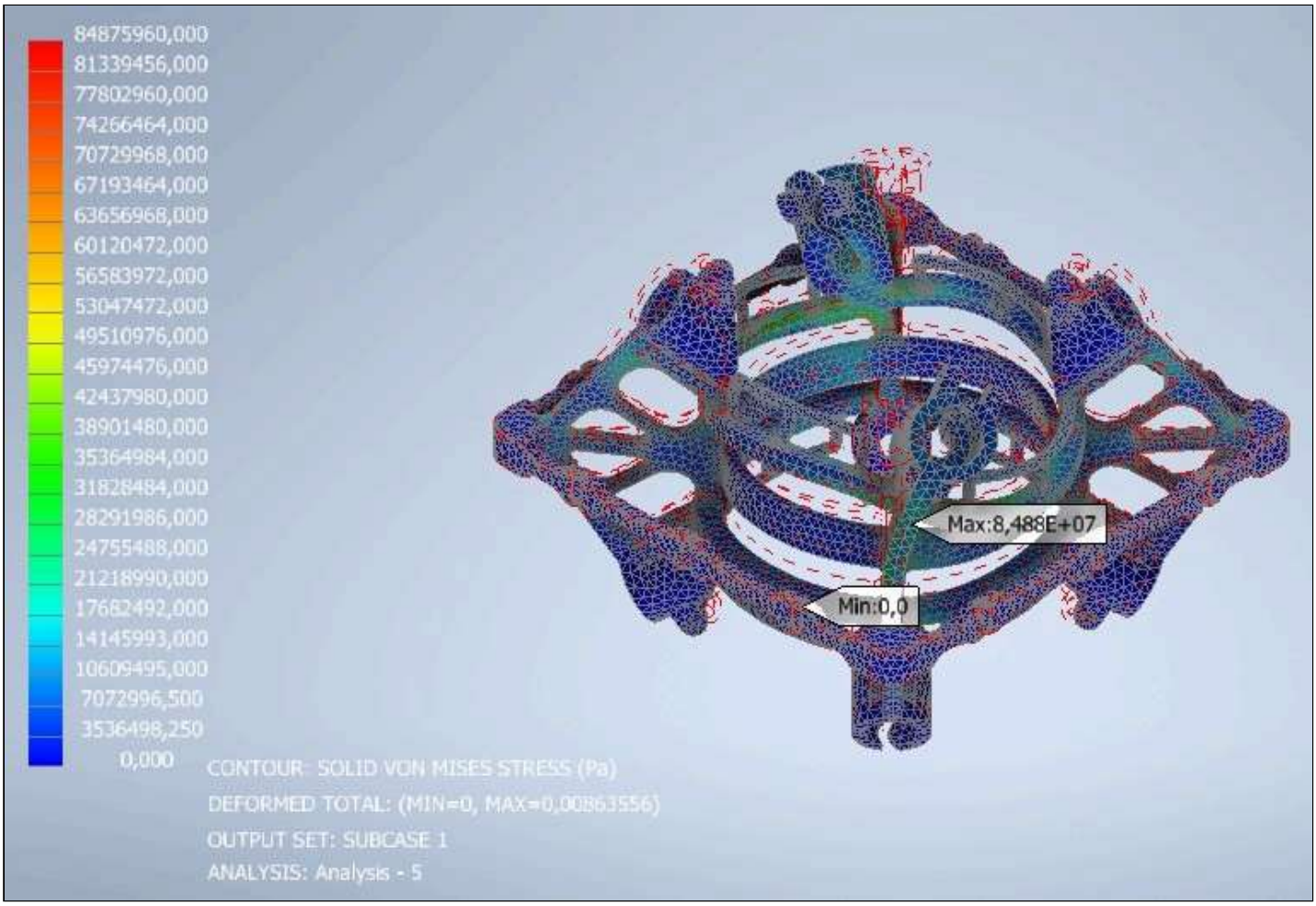

Figure 6 - OUTPUT SET: SUBCASE 1 -- DEFORMED TOTAL: (MIN=0, MAX=0,00863556) -- CONTOUR: SOLID VON MISES STRESS (Pa)

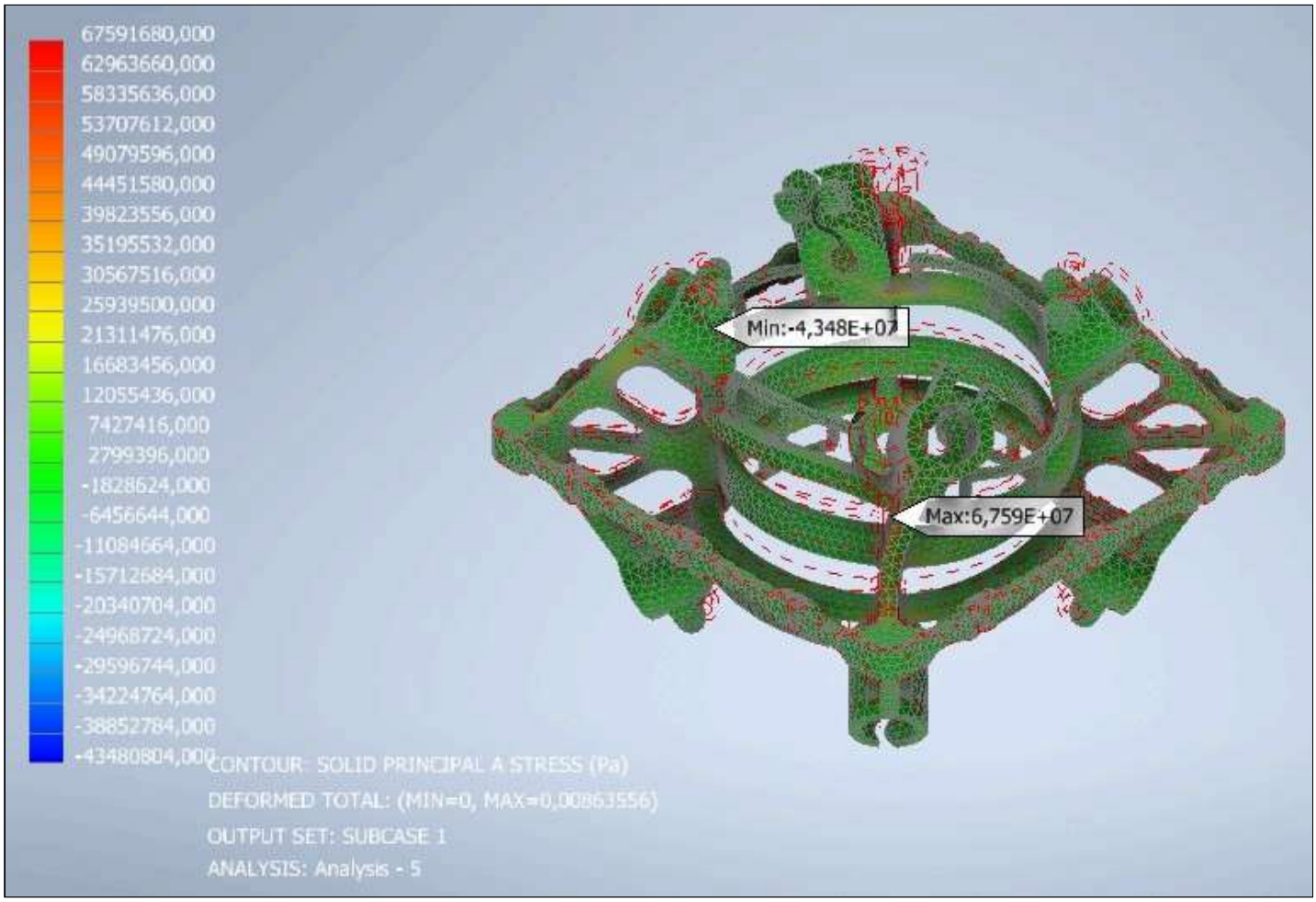

Figure 7 - OUTPUT SET: SUBCASE 1 -- DEFORMED TOTAL: (MIN=0, MAX=0,00863556) -- CONTOUR: SOLID PRINCIPAL A STRESS (Pa)

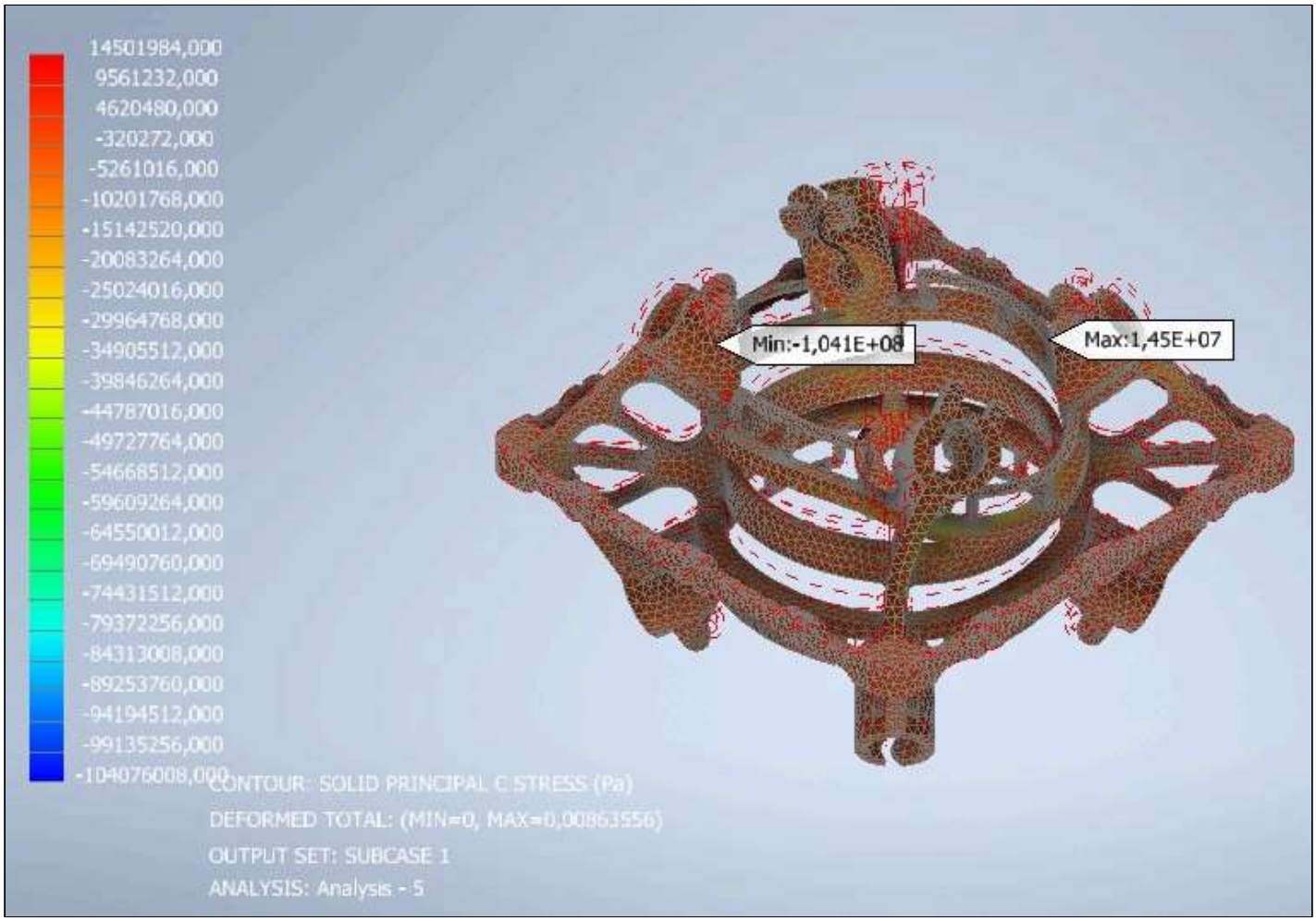

Figure 8 - OUTPUT SET: SUBCASE 1 -- DEFORMED TOTAL: (MIN=0, MAX=0,00863556) -- CONTOUR: SOLID PRINCIPAL C STRESS (Pa)

6. Conclusion:

A linear static analysis was performed using the Autodesk Nastran Version 18.2.0.35 finite element solver on the zs4ysz1gr structure. The finite element model contained mainly elements and consisted of 650328 degrees of freedom.1 loading condition was analyzed.The maximum displacement was 8,636E-03 m (load case Subcase 1)The maximum von Mises stress was 8,488E+07 (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 Forces
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.