Contents

A Analysis of the Jacks Positioning on TAN's Base

A.1 Introduction and Assumptions

Jacks should be positioned so that equal loads are taken by each of them.

In that case, each jack would support $\frac{29\,5370}{3} = 98500\ N$

Since maneuvering between the support point during installation and transfer will be important, and we are going to use 15 Tonne jacks, we can separate the jacks to the maximum distance and of the Lower Base Plate.

The loads supported by each of the jacks would then be:

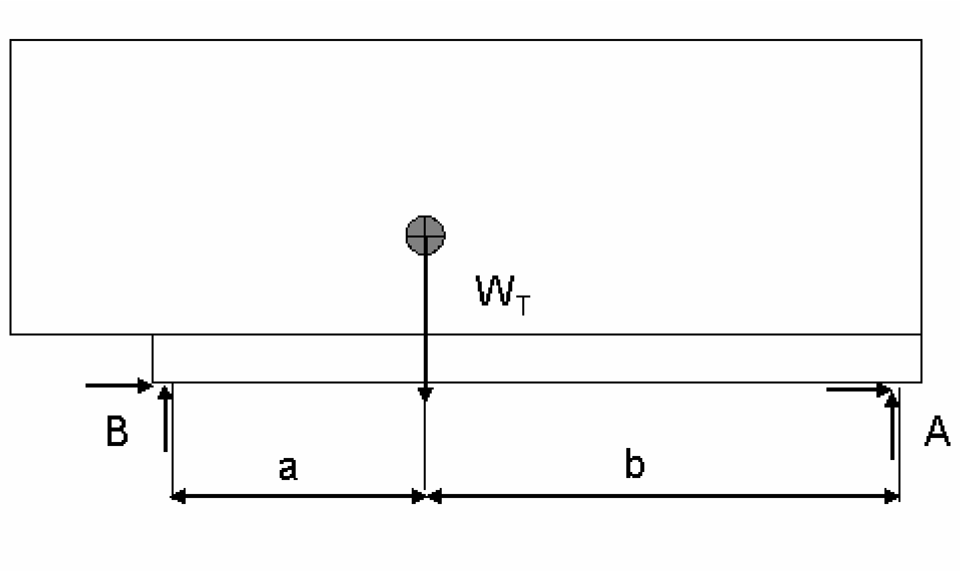


Figure A.1. Sketch of TAN's side view

$$\sum M_B = 0$$

$$W_T \cdot a = F_A \cdot (a + b)$$

$$29\,5370\ N \cdot 1\,032\ mm = F_A \cdot 2790\ mm$$

$$F_A = \frac{29\,5370 \cdot 1\,032}{2\,790} = 10\,9250{,}5\ N$$

$$\sum F = 0$$

$$F_A + 2 \cdot F_B = W_T$$

$$F_B = \frac{29\,5370 - 10\,9250{,}5}{2} = 9\,3050{,}75\ N$$

with $a = 1032mm \quad and \quad b = 1758mm$

A.2 Overturning Load

CERN's seismic safety standard requires that a device withstand 0,11 g vertical and 0,15 g horizontal acceleration without overturning.

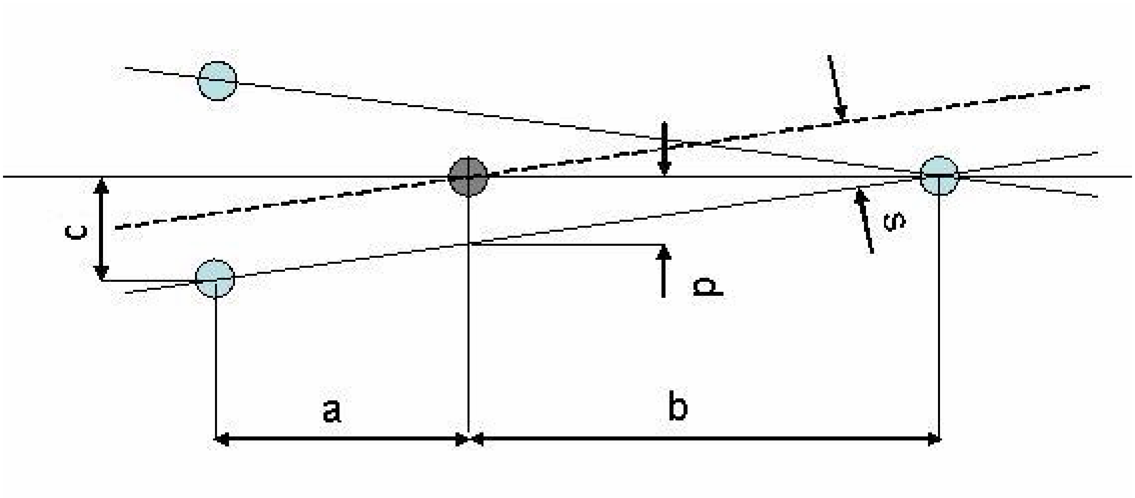


Figure A.2. Sketch of supports positioning on TAN's base (top view)

Minimum distance from c.g. to supports:

$$\frac{c}{a+b} = \frac{d}{b}$$

$$\frac{s}{d} = \frac{a+b}{\sqrt{343^2 + (a+b)^2}}$$

$$s = \frac{c \cdot b}{\sqrt{c^2 + (a+b)^2}}$$

When overturning:

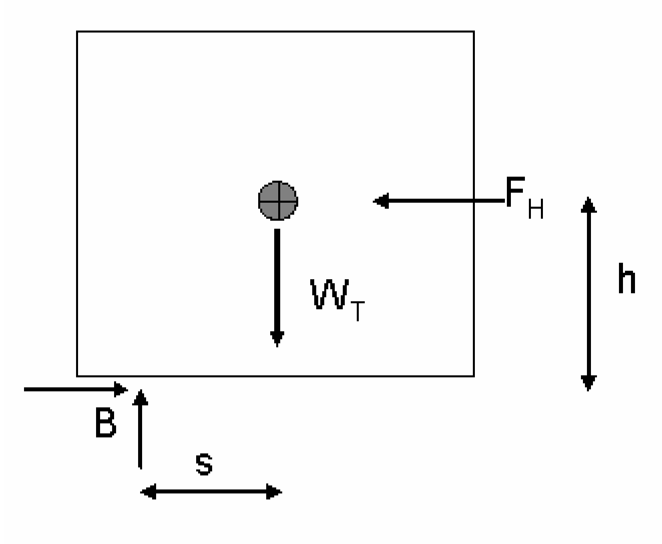


Figure A.3. Sketch of TAN's overturning (front view)

$$\sum M_B = 0$$

$$F_H \cdot h = (1 - 0{,}11) \cdot W_T \cdot s$$

$$a_H = \frac{0{,}89 \cdot s}{h}$$

The three jack's spacing proposed by Berkley Laboratory:

$2c = 486\ mm$		$s = 152{,}54\ mm$		$a_H = 0{,}24\ g$
$2c = 636\ mm$	give	$s = 199{,}10\ mm$	give	$a_H = 0{,}31\ g$
$2c = 786\ mm$		$s = 245{,}21\ mm$		$a_H = 0{,}38\ g$

All the three proposed jacks spacings meet the CERN seismic safety standard of 0,11 g vertical and 0,15 g horizontal acceleration.

The definitive jacks positioning will finally be:

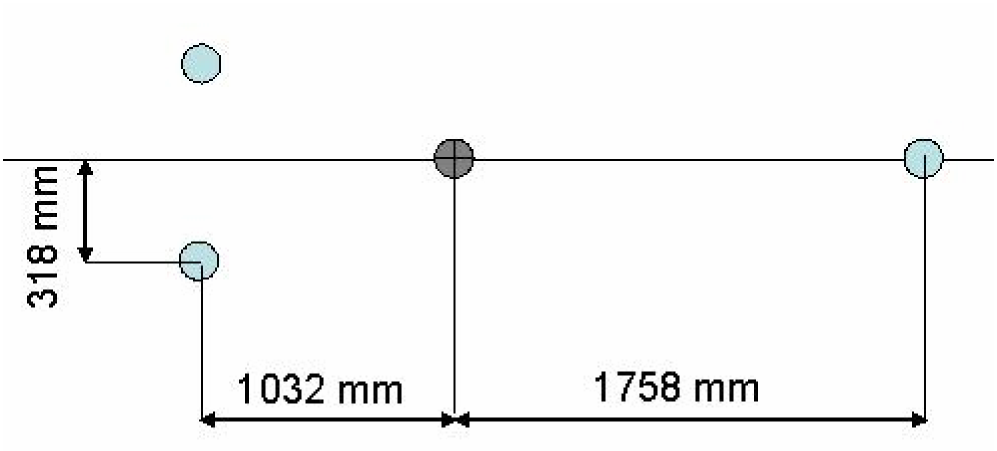


Figure A.4. Sketch of the final position of jacks on TAN's base (top view)

A.3 TAN Jack Floor Plates

The three support jacks will rest in floor plates. The minimum size of these floor plates will have to be:

$$P_{MAX} = \frac{F_{JACK}}{A_{MIN}}$$

Where:

P_{MAX} = 0, 5 MPa (Stainless Steel 304 L Yield Strength)

$$A_{MIN1} = \frac{10\,925{,}5Kg \cdot 9{,}8\, {}^{N}\!/_{Kg}}{0{,}5 \cdot 10^{6}\, {}^{N}\!/_{m^{2}}} = 0{,}22\ m^{2}$$

$$A_{MIN2} = \frac{9\,305{,}75Kg \cdot 9{,}8\, {}^{N}\!/_{Kg}}{0{,}5 \cdot 10^{6}\, {}^{N}\!/_{m^{2}}} = 0{,}19\ m^{2}$$

Jack 1 will rest on a plate:

500 mm x 475 mm > 0, 22 mm^2

Jacks 2 & 3 will rest on a plate:

950 mm x 475 mm > 0, 38 mm^2

B Analysis of the horizontal handling of the TAN's Vacuum Chamber

B.1 Introduction and Assumptions

In order to ensure that the critical areas of the Vacuum Chambers are not overstressed during testing and assembling a horizontal analysis using two support points has been performed.

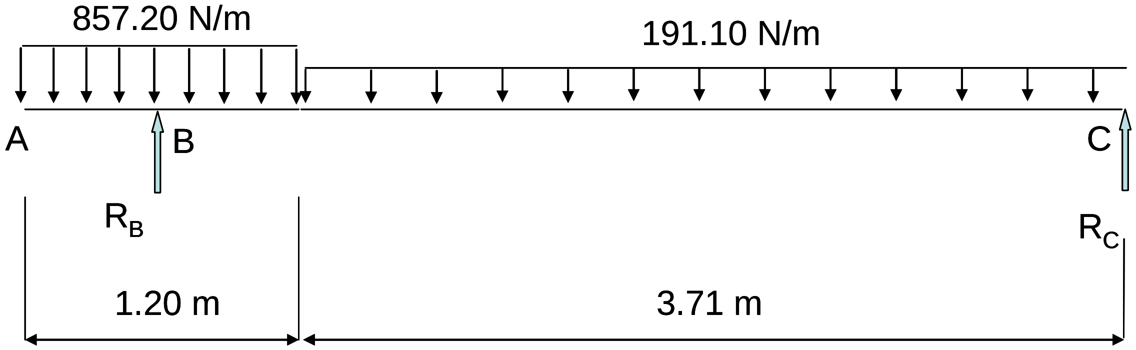
By recommendation of E. Hoyer (Berkley University) the maximum allowable tensile stress in the electron beam welds is 5000 psi and in the twin beamtubes 3100 psi. Therefore, for this study, I will assume the more conservative value of 3100 psi (2180 N/cm^2) has the maximum allowable stress of the whole chamber.

The Vacuum Chamber can be modeled by two different parts: A large end and a twin beamtube. The moments of inertia of the sections were determined to be $I_{Large\ end}$=1794 cm^4 and $I_{Beamtube}$= 84.84 cm^4. The large end weight is approximately 85.72 Kg/m and the twin beamtube weight is 19.11 Kg/m. The large end exterior radius is 11.50 cm and the twin beamtube exterior radius is 3.16 cm.

It is assumed that stresses in the large end can be ignored since the moment of inertia is so large. 2180 N/cm^2 maximum allowable stress in the beamtubes equates to a 585.29 N-m maximum allowable moment.

B.2 The first model

A first model consists of the two sections of the Vacuum chamber with their own weights with two support points (one at the c.g. of the large end the other one at the extremity of the twin beamtube):



Calculating the reactions R_A and R_B for this model:

$$\sum F = 0$$

$$857.20 \cdot 1.20 + 191.10 \cdot 3.71 = R_B + R_C$$

$$1028.64 + 708.68 = R_B + R_C$$

$$1737.31 = R_B + R_C$$

$$\sum M_B = 0$$

$$708.68 \cdot \left(\frac{3.71}{2} + 0.60 \right) = R_C \cdot (3.71 + 0.60)$$

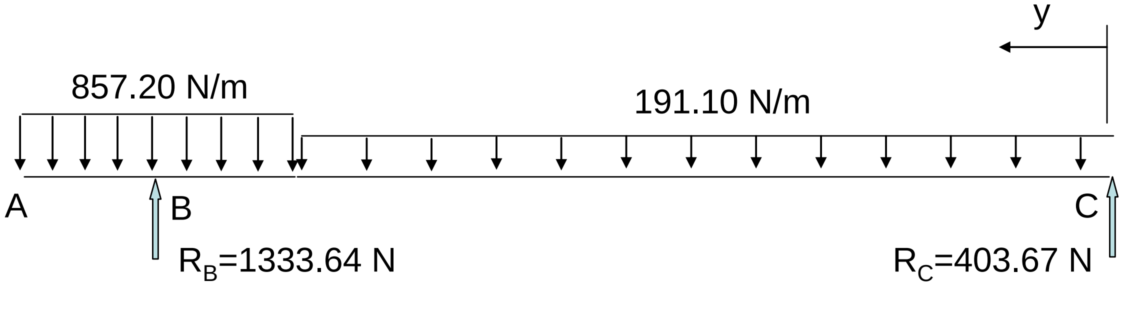
$$708.68 \cdot 2.455 = 4.31 \cdot R_C$$

$$R_C = \frac{708.68 \cdot 2.455}{4.31}$$

$$R_C = 403.67 \ N$$

$$R_B = 1333.64 \ N$$

Since the maximum bending moment is located where the shear is zero we first locate that point:



$$T(y) = 403.67 - 191.10 \cdot y$$

$T(y)=0 \rightarrow \quad y=2.11\,m$ (From the right end → on the twin beamtube part)

Calculating now the value of the bending moment in that point:

$$M(y)=403.67y-191.10\frac{y^2}{2}$$

$$M(2.11)=852.69-425.40=427.30\ N\cdot m < M_{MAX}=585.29\ N\cdot m$$

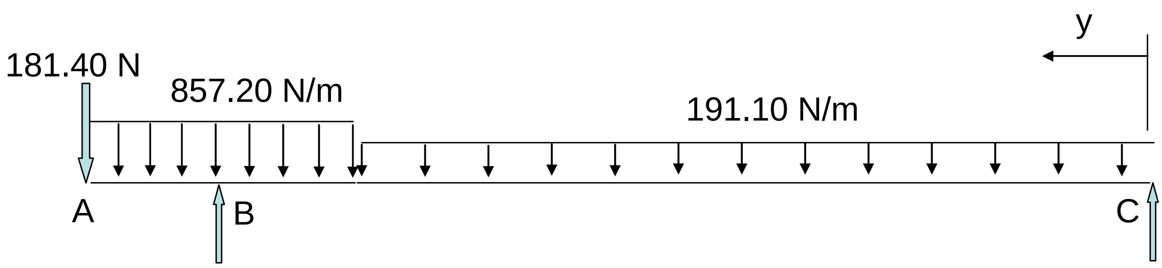
Resulting in a stress of:

$$42730\ N\cdot cm\cdot\frac{3.16\ cm}{84.84\ cm^4}=1591.55\ {}^{N}\!/_{cm^2} < S_{MAX}=2180\ {}^{N}\!/_{cm^2}$$

These results are acceptable since they are below the moments and stresses allowables.

B.3 The second model

The second model includes the action of the flange adapter (181.4 N) at the left end of the large end section:



Recalculating the reactions R_A and R_B for this 2nd model:

$$\sum F=0$$

$$857.20\cdot 1.20+191.10\cdot 3.71+181.4=R_B+R_C$$

$$1028.64+708.68=R_B+R_C$$

$$1918.71=R_B+R_C$$

$$\sum M_B = 0$$

$$708.68 \cdot \left(\frac{3.71}{2} + 0.60 \right) - 181.4 \cdot 0.6 = R_C \cdot (3.71 + 0.60)$$

$$708.68 \cdot 2.455 - 108.84 = 4.31 \cdot R_C$$

$$R_C = \frac{1739.81 - 108.84}{4.31}$$

$$R_C = 378.42\ N$$

$$R_B = 1540.29\ N$$

To locate the maximum bending moment I recalculate the point where the shear is zero:

$$T(y) = 378.42 - 191.10 \cdot y$$

$$T(y) = 0 \rightarrow \quad y = 1.98\ m$$

Calculating now the value of the bending moment in that point:

$$M(y) = 378.42 y - 191.10 \frac{y^2}{2}$$

$$M(1.98) = 749.27 - 374.64 = 374.67\ N \cdot m < M_{MAX} = 585.29\ N \cdot m$$

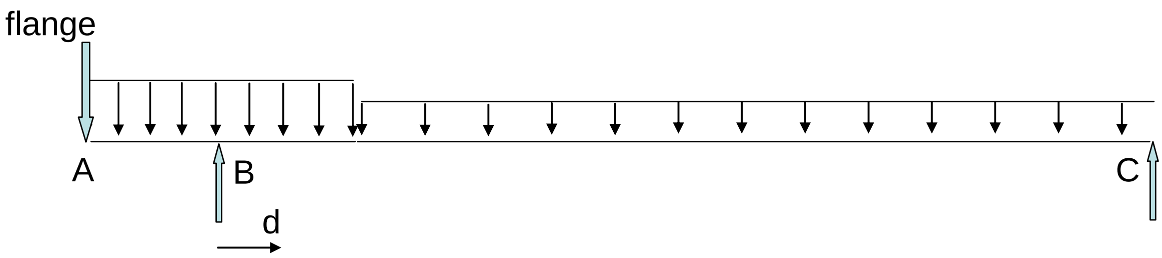
Resulting in a stress of:

$$37467\ N \cdot cm \cdot \frac{3.16\ cm}{84.84\ cm^4} = 1395.51\ {N}/{cm^2} < S_{MAX} = 2180\ {N}/{cm^2}$$

The stresses of this model are slightly smaller since the action of the flange reduces the moments in the twin beamtube.

B.4 The general model

In order to assure even better the lifting and give a tolerance for the handling procedure, we can study how the reactions and moments act when moving the support point B inside the large end part:



Finding the relation between the maximal moment (M) and the distance between the old point B and the new one (d):

$$T(y)=0 \rightarrow R_C - 191.10y = 0$$

$$M(y) = M_{\max} \rightarrow R_C \cdot y - 191.10 \cdot \frac{y^2}{2} = M_{\max}$$

$$191.10\frac{y^2}{2} = M_{\max} \rightarrow y = \sqrt{\frac{2M_{\max}}{191.10}}$$

$$R_C = 191.10 \cdot \sqrt{\frac{2M_{\max}}{191.10}} = \sqrt{382.2 \cdot M_{\max}} = 19.55 \cdot (M_{\max})^{1/2}$$

$$\sum M_{B'} = 0$$

$$181.4 \cdot (0.6 + d) + 1028.54 \cdot d + R_C \cdot (4.31 - d) - 708.68 \cdot (2.455 - d) = 0$$

$$1918.72 \cdot d + R_C \cdot (4.31 - d) = 1630.97$$

$$-19.55 \cdot d \cdot M_{MAX}^{1/2} + 84.26 \cdot M_{MAX}^{1/2} + 1918.72 \cdot d - 1630.97 = 0$$

$$M_{MAX}^{1/2} \cdot (-19.55 \cdot d + 84.26) = -1918.72 \cdot d + 1630.97$$

$$M_{MAX} = \left(\frac{-1918.72 \cdot d + 1630.97}{19.55 \cdot d + 84.26} \right)^2$$

The Table B.1 shows the values of the Maximal Bending Moment, Maximal Stress and the reactions for different positions of the support B:

d [m]	M_{MAX} [N m]	σ_{max} [N/cm^2]	R_c [N]	R_B [N]
-0,60	840,08	3 129,01	566,64	1 352,07
-0,50	758,80	2 826,26	538,53	1 380,18
-0,40	678,47	2 527,06	509,23	1 409,48
-0,30	599,44	2 232,71	478,65	1 440,06
-0,20	522,13	1 944,76	446,72	1 471,99
-0,10	447,02	1 665,00	413,34	1 505,37
0,00	374,67	1 395,50	378,42	1 540,29
0,10	305,72	1 138,70	341,83	1 576,88
0,20	240,94	897,43	303,46	1 615,25
0,30	181,22	674,99	263,18	1 655,53
0,40	127,60	475,28	220,84	1 697,87
0,50	81,30	302,81	176,28	1 742,43
0,60	43,75	162,95	129,31	1 789,40

Table B.1. Values of the Maximal Bending Moment, stresses and reactions in function of the position of the support point B

From distances d=0.10 to d=0.6 the large end part enters in its transition segment. In this transition segment, where a special particle detector will be placed, we are not allowed to set a support (delimitated by the vertical red line x = 0.10).

As we could imagine and we can see in Table B.2 the incidence of the reactions on the support points B and C does not compromise the structure since the tangential stresses are so low:

d [m]	R_c [N]	σ_B (2T/A) (N)	R_B [N]	σ_C (2T/A) (N)
-0,60	566,64	55,91	1 352,07	88,34
-0,50	538,53	53,14	1 380,18	90,18
-0,40	509,23	50,24	1 409,48	92,09
-0,30	478,65	47,23	1 440,06	94,09
-0,20	446,72	44,08	1 471,99	96,18
-0,10	413,34	40,78	1 505,37	98,36
0,00	378,42	37,34	1 540,29	100,64
0,10	341,83	33,73	1 576,88	103,03
0,20	303,46	29,94	1 615,25	105,54
0,30	263,18	25,97	1 655,53	108,17
0,40	220,84	21,79	1 697,87	110,94
0,50	176,28	17,39	1 742,43	113,85
0,60	129,31	12,76	1 789,40	116,92

Table B.2. Values of the tangential stresses induced by the reactions on the support points

Choosing the support B to be situated 0.65 meters from the left end of the Vacuum Chamber (d=0.05) will induce low and acceptable stresses to the structure and will give a enough tolerance (± 50 mm) for the handling team. The values of the reactions, maximal moment and stresses will be:

d [m]	M_{MAX} [N m]	σ_{max} [N/cm^2]	R_c [N]	σ_B [N]	R_B [N]	σ_C [N]
0,05	339,72	1 265,35	360,34	35,55	1 558,37	101,82

Table B.3. Values of the reactions, maximal moments and stresses for the final position of support B

Two support points will be settled:

Support B: situated 650 mm. from the left end of the Vacuum Chamber (with a tolerance of ± 50 mm)

Support C: situated at the right end of the Vacuum Chamber (a little inboard ± 50 mm)

C The Vehicle for TANs Installation (VTI)

C.1 Sketches of a possible VTI configuration:

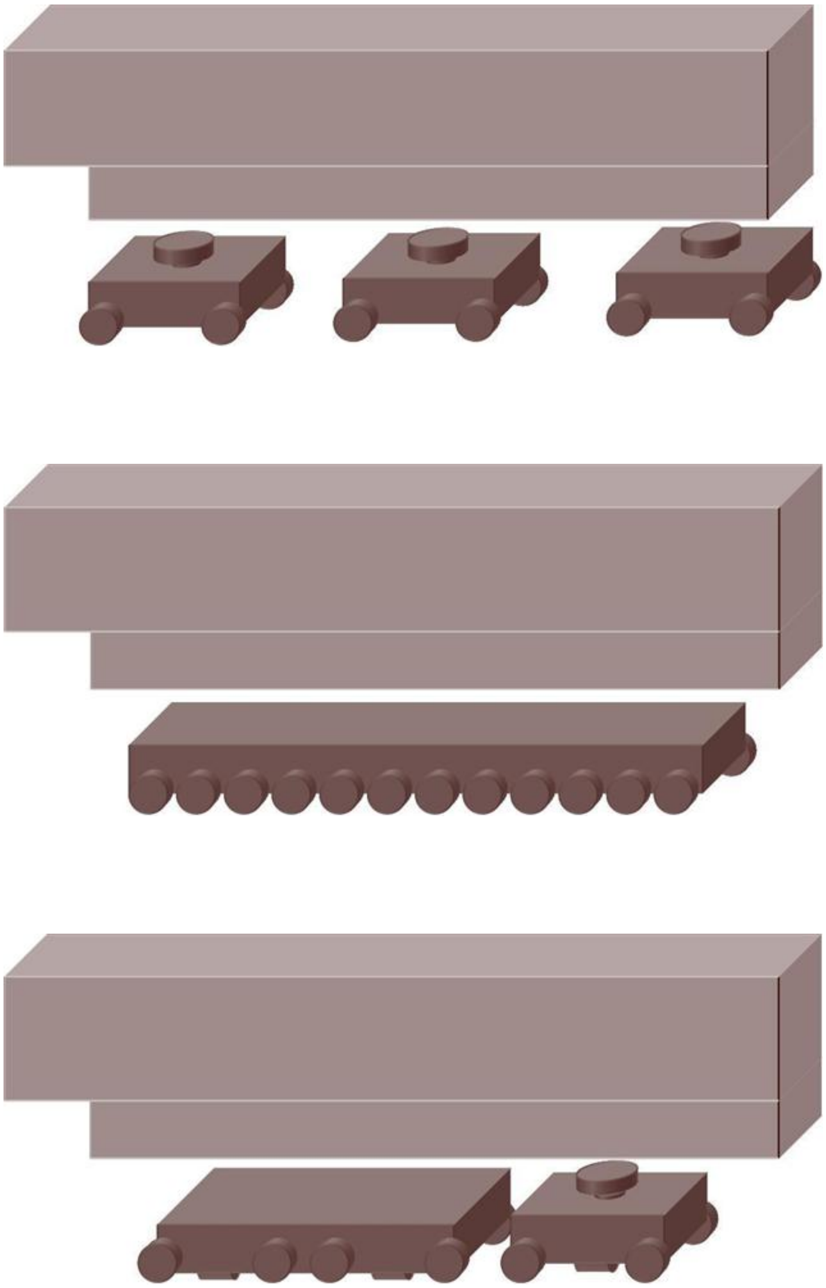


Figure C.1. Sketches of a possible VTI

C.2 Sketches of a possible installation procedure:

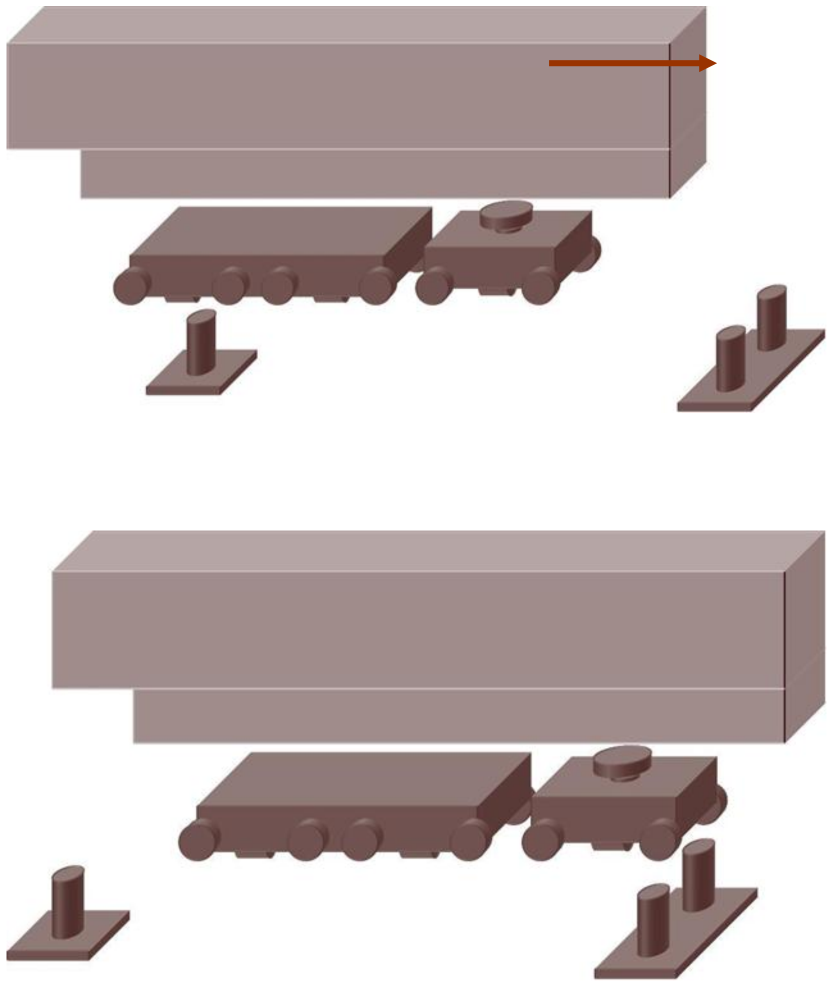


Figure C.2. Transport of the TAN to the final installation

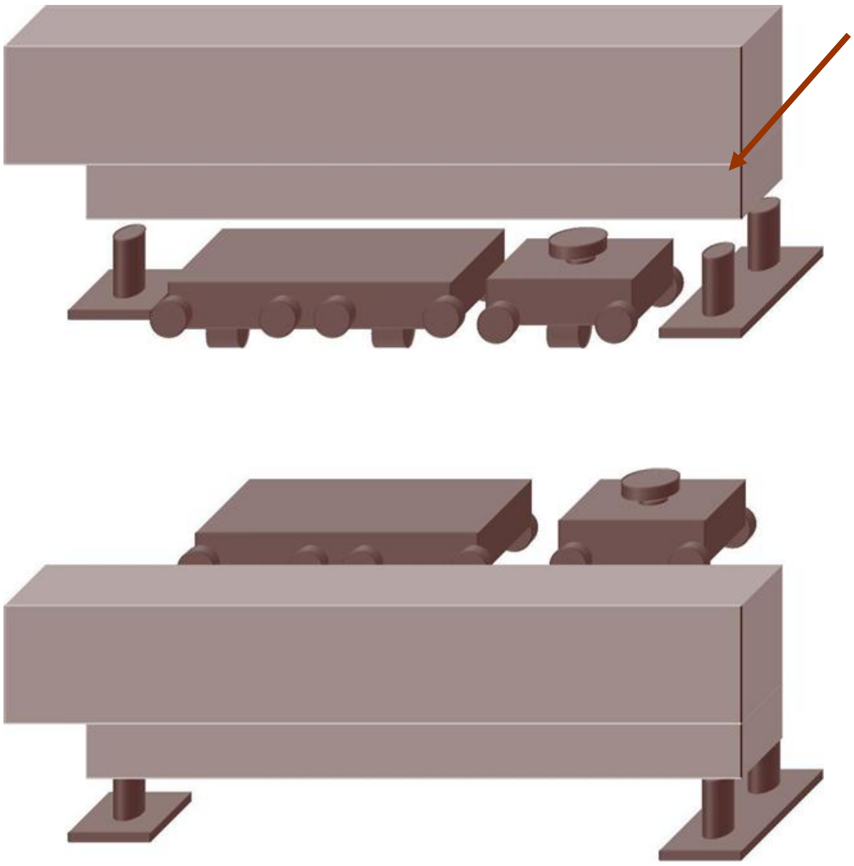


Figure C.3. Transfer of the TAN onto the support jacks

C.3 Applicable Documents

C.3.1 CERN standards

The following standards, in order of priority, are applicable for the design and fabrication of the VTI. These documents can be found in the url: http://lhc-proj-qawg.web.cern.ch/lhc-proj-qawg/CD-ROM-v4-0/Contents.pdf.

- Safety Regulations Applicable To The Work Of Contractors At CERN [CERN/TIS-GS/98-10] (1998).
- Special conditions for the operation and/or maintenance of CERN equipment and installations. [CERN/FI/120] (1995).
- Contractors and their staff: Access to and activities on the CERN site. [DSU-DO/RH/1845] (1995)
- Lifting equipment. [CERN Safety Code D1] (1997).
- Pressure equipment. [CERN Safety Code D2] (1998).
- Electrical Safety Code. [CERN Safety Code C1] (1996).
- Emergency stops. [CERN Safety Instructions IS 5] (1985).
- Criteria for the selection of electrical cables and equipment with respect to fire safety and radiation resistance. [CERN Safety Instructions IS 23] (1992/93).
- The use of plastics and other non-metallic materials at CERN with respect to fire safety and radiation resistance. [CERN Safety Instructions IS 41] (1995).

C.3.2 Quality Assurance Provisions

The Design and Construction of the VTI must also adhere to a documented quality assurance program that fulfils all the requirements described in the Quality Assurance Plan for the LHC Project.

The list of relevant topics covered by the LHC Quality Assurance Plan, together with the corresponding documents, is given below in Table C.1. These documents can also be found in the url: http://lhc-proj-qawg.web.cern.ch/lhc-proj-qawg/CD-ROM-v4-0/Contents.pdf.

Topic	Document Title	Doc. Number
Policy and Organisation	Quality Assurance Policy and Organisation	LHC-PM-QA-100.00
	Glossary, Acronyms, Abbreviations	LHC-PM-QA-203.00
Planning	Planning and Scheduling Requirements for Institutes, Contractors and Suppliers	LHC-PM-QA-301.01
Design	Quality Assurance Categories	LHC-PM-QA-201.00
	Design Process and Control	LHC-PM-QA-307.00
	Drawing Management and Control	LHC-PM-QA-305.00
	Drawing Process-External Drawings	LHC-PM-QA-306.00
Change Control	Configuration Management - Change Process And Control	LHC-PM-QA-304.00
Manufacturing and Inspection	Manufacturing and Inspection of Equipment	LHC-PM-QA-309.00
	Handling of Non-conforming Equipment	LHC-PM-QA-310.00
	LHC Part Identification	LHC-PM-QA-206.00

Table C.1 - LHC QAP topics and documents

D Simulations of the NEG Coating Lifting Assembly

D.1 Model 1

ANSYS Report

Author

OSCAR FERNANDEZ

Subject

Horizontal Analysis of the two beams + cut (Acting only gravity)

Prepared For

NEG Coating for TAN VAcuum Chamber

Project Created

Tuesday, November 23, 2004 at 4:53:04 PM

Project Last Modified

Wednesday, November 24, 2004 at 11:19:46 AM

Report Created

Thursday, May 26, 2005 at 4:52:11 PM

Software Used

ANSYS 8.1

Database

G:\Divisions\EST\Groups\IC\LC\Oscar\DOCUMENTOS\PROYECTO\TAN\Pruebas vacio\ansys workbench\dosvigas_y_placa_cortada_extremo(nuevos_agujeros)_solo gravedad.dsdb

D.1.1 Summary

This report documents design and analysis information created and maintained using the ANSYS® engineering software program. Each scenario listed below represents one complete engineering simulation.

Scenario 1

- Based on the Autodesk® Mechanical Desktop® assembly *"G:\Divisions\EST\Groups\IC\LC\Oscar\DOCUMENTOS\PROYECTO\TAN\Pruebas vacio\ansys workbench\dosvigas_y_placa_cortada_extremo(nuevos_agujeros).dwg"*.
- Considered the effect of body-to-body contact, acceleration and structural supports.
- Calculated structural results.
- No convergence criteria defined.
- No alert criteria defined.
- See Scenario 1 below for supporting details and Appendix A1 for corresponding figures.

D.1.2 Introduction

The ANSYS CAE (Computer-Aided Engineering) software program was used in conjunction with 3D CAD (Computer-Aided Design) solid geometry to simulate the behavior of mechanical bodies under thermal/structural loading conditions. ANSYS automated FEA (Finite Element Analysis) technologies from ANSYS, Inc. to generate the results listed in this report.

Each scenario presented below represents one complete engineering simulation. The definition of a simulation includes known factors about a design such as material properties per body, contact behavior between bodies (in an assembly), and types and magnitudes of loading conditions. The results of a simulation provide insight into how the bodies may perform and how the design might be improved. Multiple scenarios allow comparison of results given different loading conditions, materials or geometric configurations.

Convergence and alert criteria may be defined for any of the results and can serve as guides for evaluating the quality of calculated results and the acceptability of values in the context of known design requirements.

- *Solution history* provides a means of assessing the quality of results by examining how values change during successive iterations of solution refinement. *Convergence criteria* sets a specific limit on the allowable change in a result between iterations. A result meeting this criteria is said to be "converged".
- *Alert criteria* define "allowable" ranges for result values. Alert ranges typically represent known aspects of the design specification.

The discussions below follow the organization of information in the ANSYS "Explorer" user interface. Each scenario corresponds to a unique branch in the Explorer "Outline". Names emphasized in *"double quotes"* match preferences set in the user interface.

All values are presented in the *"SI Metric (m, kg, N, °C, s, V, A)"* unit system.

Notice

Do not accept or reject a design based solely on the data presented in this report. Evaluate designs by considering this information in conjunction with experimental test data and the practical experience of design engineers and analysts. A quality approach to engineering design usually mandates physical testing as the final means of validating structural integrity to a measured precision.

D.1.3 "Model"

"Model" obtains geometry from the Autodesk® Mechanical Desktop® assembly *"G:\Divisions\EST\Groups\IC\LC\Oscar\DOCUMENTOS\PROYECTO\TAN\Pruebas vacio\ansys workbench\dosvigas_y_placa_cortada_extremo(nuevos_agujeros).dwg"*.

- The bounding box for all positioned bodies in the model measures 0.2 by 0.23 by 8.5 m along the global x, y and z axes, respectively.
- The model has a total mass of 146.84 kg.
- The model has a total volume of 5.3×10^{-2} m³.

Bodies

Name	**Material**	**Bounding Box (m)**	**Mass (kg)**	**Volume (m³)**	**Nodes**	**Elements**
"PART1_1"	*"Aluminum Alloy"*	0.15, 0.15, 2.0	28.99	1.05×10^{-2}	2823	540
"PART2_1"	*"Aluminum Alloy"*	0.1, 0.22, 6.5	71.25	2.57×10^{-2}	14779	7734
"Part"	*"Aluminum Alloy"*	0.2, 0.01, 7.78	43.06	1.55×10^{-2}	10946	1918
"Part 2"	*"Aluminum Alloy"*	0.2, 0.01, 0.64	3.55	1.28×10^{-3}	141	16

Body Groupings

Name	**Body Names**	**Bounding Box (m)**	**Mass (kg)**	**Volume (m³)**	**Nodes**	**Elements**
"PART3_1"	Part , Part 2	200.0, 10.0, 8,500.0	46.61	1.68×10^{7}	11,087.0	1,934.0

D.1.4 Contact

- *"Contact"* uses a tolerance of 0.0 for automatic detection.

Contact Conditions

Name	**Type**	**Associated Bodies**	**Scope**	**Normal Stiffness**	**Scope Mode**	**Behavior**	**Formulation**	**Thermal Conductance**	**Pinball Region**
"Contact Region"	Bonded	*"PART2_1"* and *"PART1_1"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 2"	Bonded	*"Part"* and *"PART1_1"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 3"	Bonded	*"Part"* and *"PART2_1"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 4"	Bonded	*"Part 2"* and *"PART2_1"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled

D.1.5 Mesh

- *"Mesh"*(Figure A1.1) , associated with *"Model"* has an overall relevance of 0.
- *"Mesh"* contains 28689 nodes and 10208 elements.

No mesh controls specified.

D.1.6 "Environment"

"Environment"(Figure A1.2) contains all loading conditions defined for *"Model"* in this scenario.

Standard Earth Gravity - Standard Earth Gravity

- Magnitude: 9.81 m/s²
- Vector: [0.0 m/s² x, 9.81 m/s² y, 0.0 m/s² z] in the global coordinate system

The following tables list local loads and supports applied to specific geometry.

D.1.7 Structural Supports

Structural Supports

Name	**Type**	**Reaction Force**	**Reaction Force Vector**	**Reaction Moment**	**Reaction Moment Vector**	**Associated Bodies**
"Fixed Support"	Fixed Edge	5,501.4 N	[-0.17 N x, 726.14 N y, 5,453.26 N z]	8.66 N·m	[0.0 N·m x, 8.58 N·m y, -1.15 N·m z]	*"PART2_1"*
"Fixed Support 2"	Fixed Edge	5,498.34 N	[0.17 N x, 702.59 N y, -5,453.26 N z]	0.32 N·m	[1.51×10^{-13} N·m x, -0.31 N·m y, 0.11 N·m z]	*"PART1_1"*

D.1.8 "Solution"

"Solution" contains the calculated response for *"Model"* given loading conditions defined in *"Environment"*.

It was selected that the program would choose the solver used in this solution.

D.1.9 Structural Results

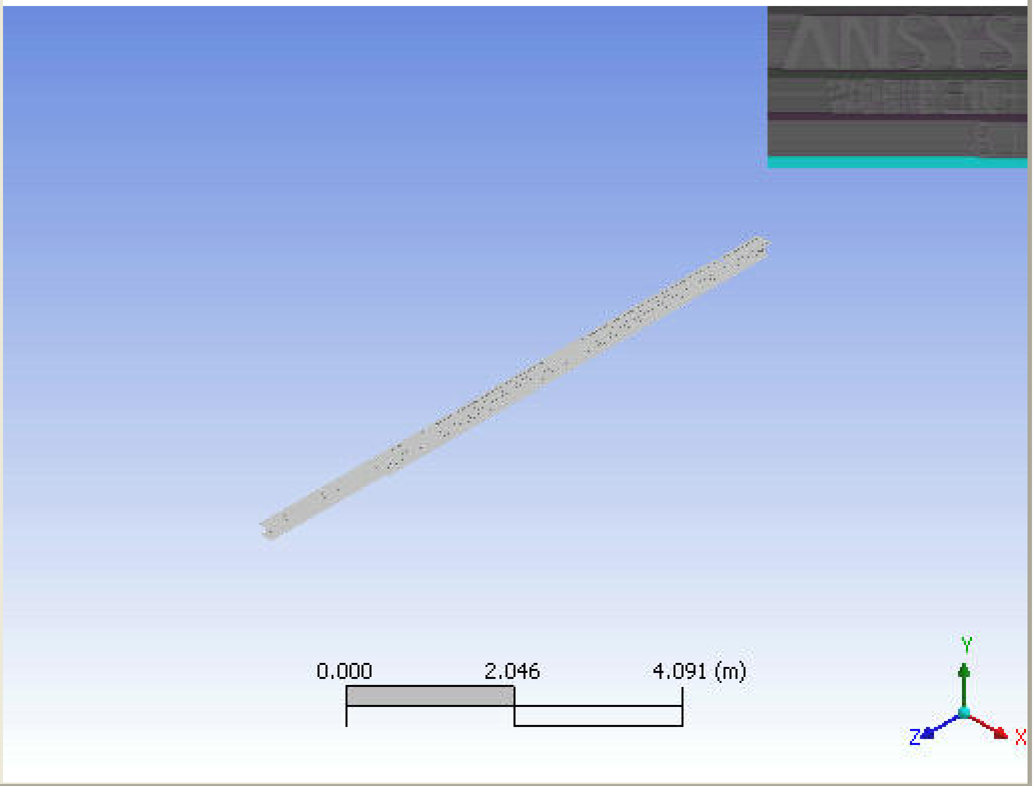
Values

Name	**Figure**	**Scope**	**Minimum**	**Maximum**	**Alert Criteria**
"Equivalent Stress"	A1.3	All Bodies In *"Model"*	3,286.31 Pa	7.63×10^{6} Pa	None
"Total Deformation"	A1.4	All Bodies In *"Model"*	0.0 m	1.81×10^{-3} m	None

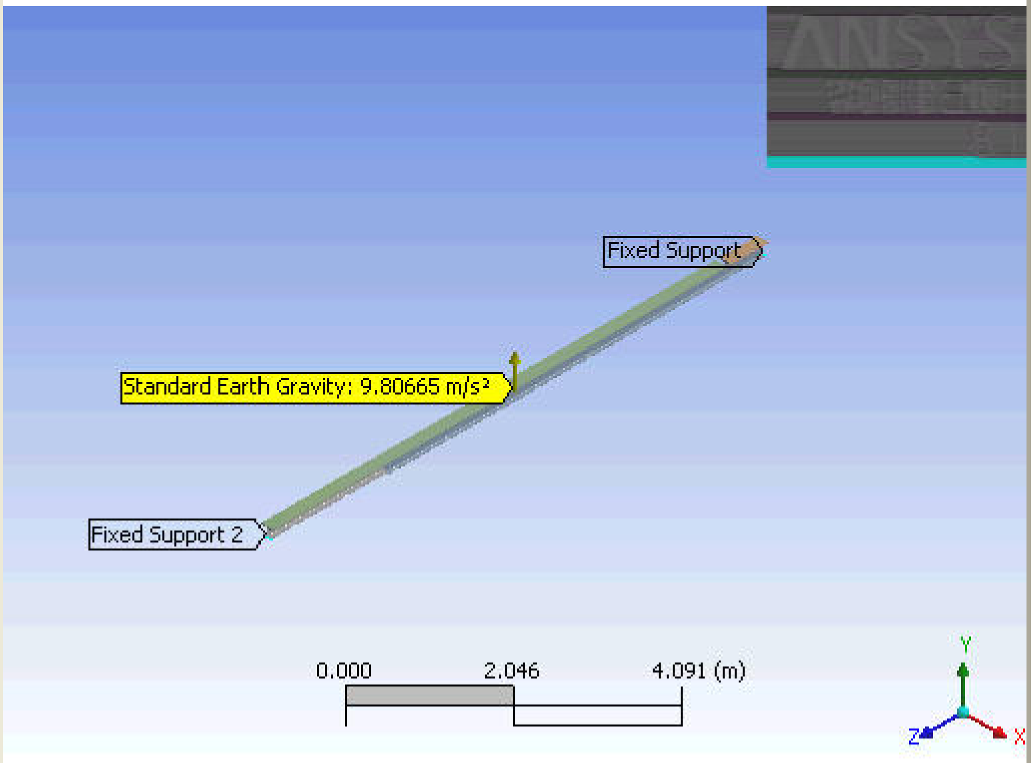
- Convergence tracking not enabled.

D.1.10 Figures

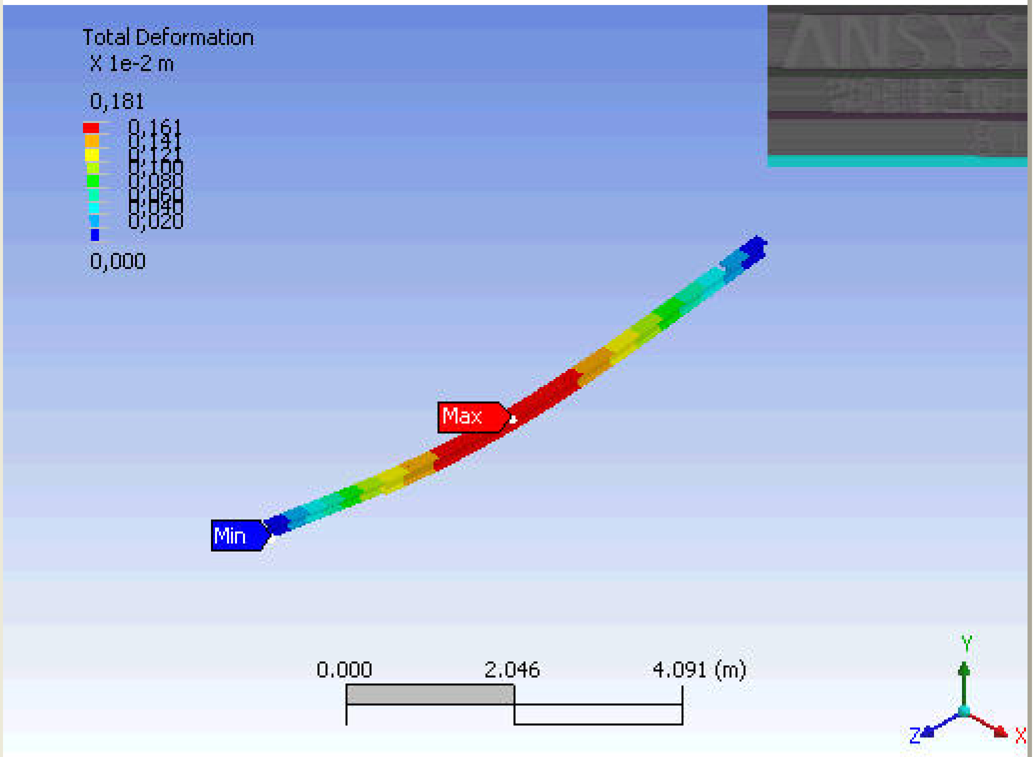
***"Mesh"* Geometry**



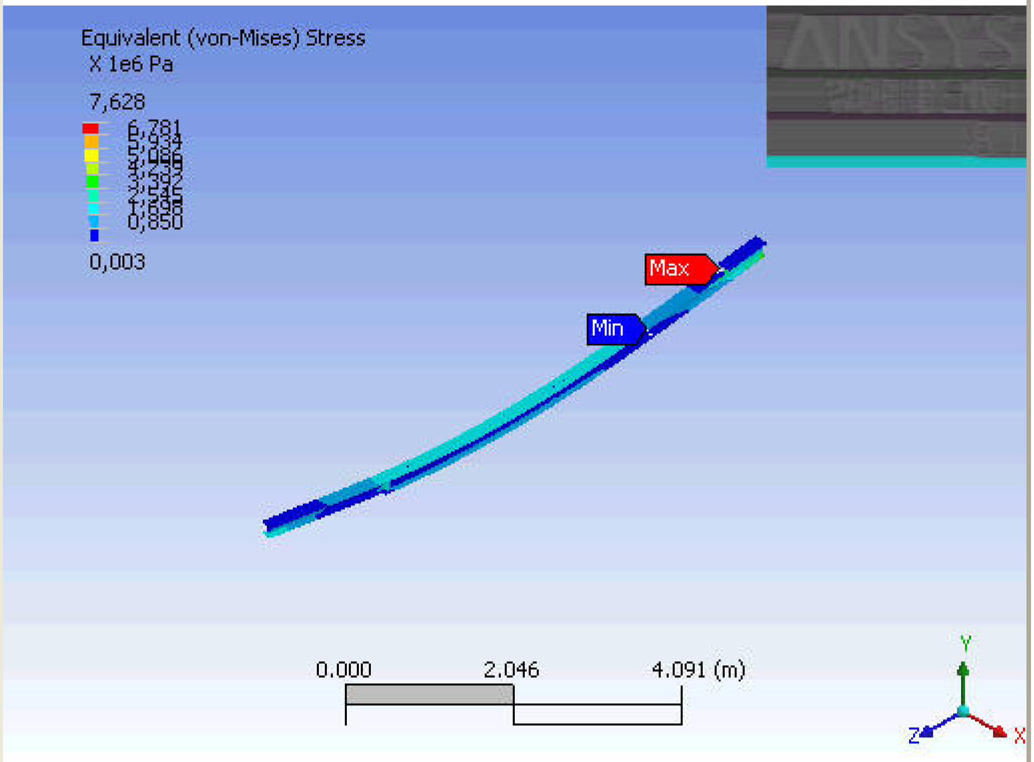
Environment" Geometry



"Total Deformation" **Contours**



"*Equivalent Stress*" Contours



D.1.11 Definition of "Aluminum Alloy"

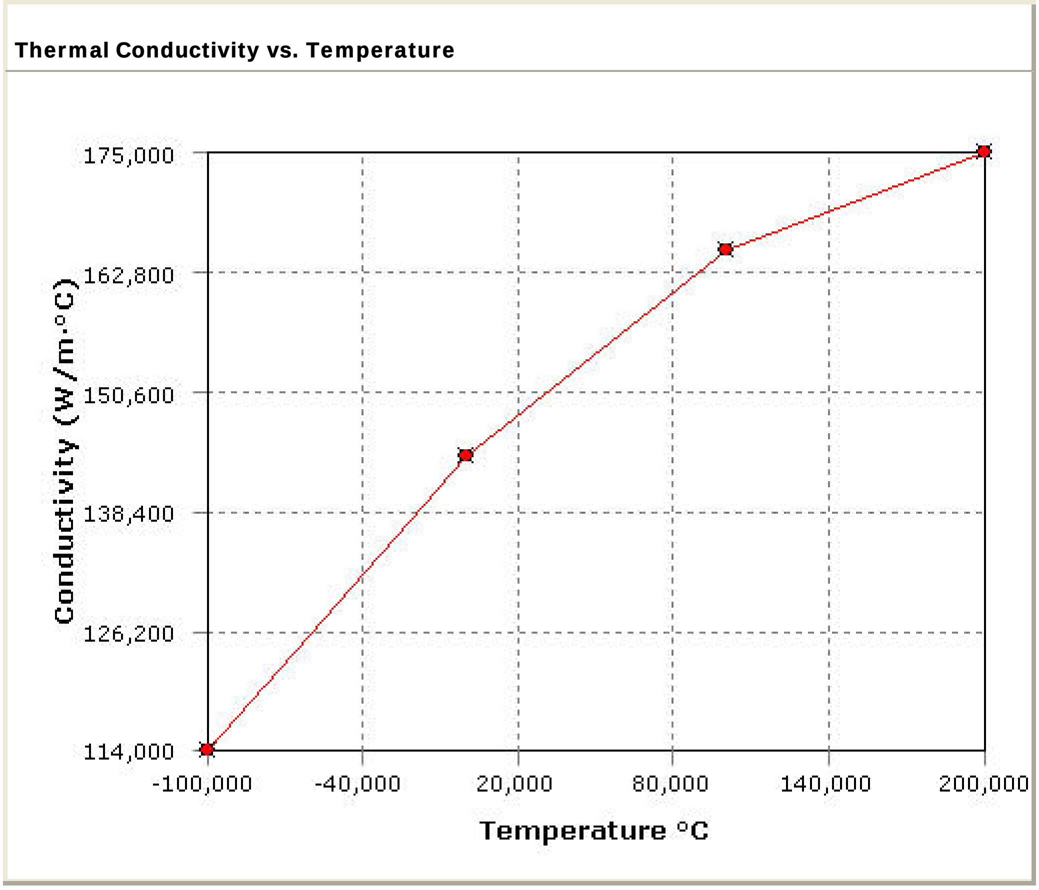
***"Aluminum Alloy"* Properties**

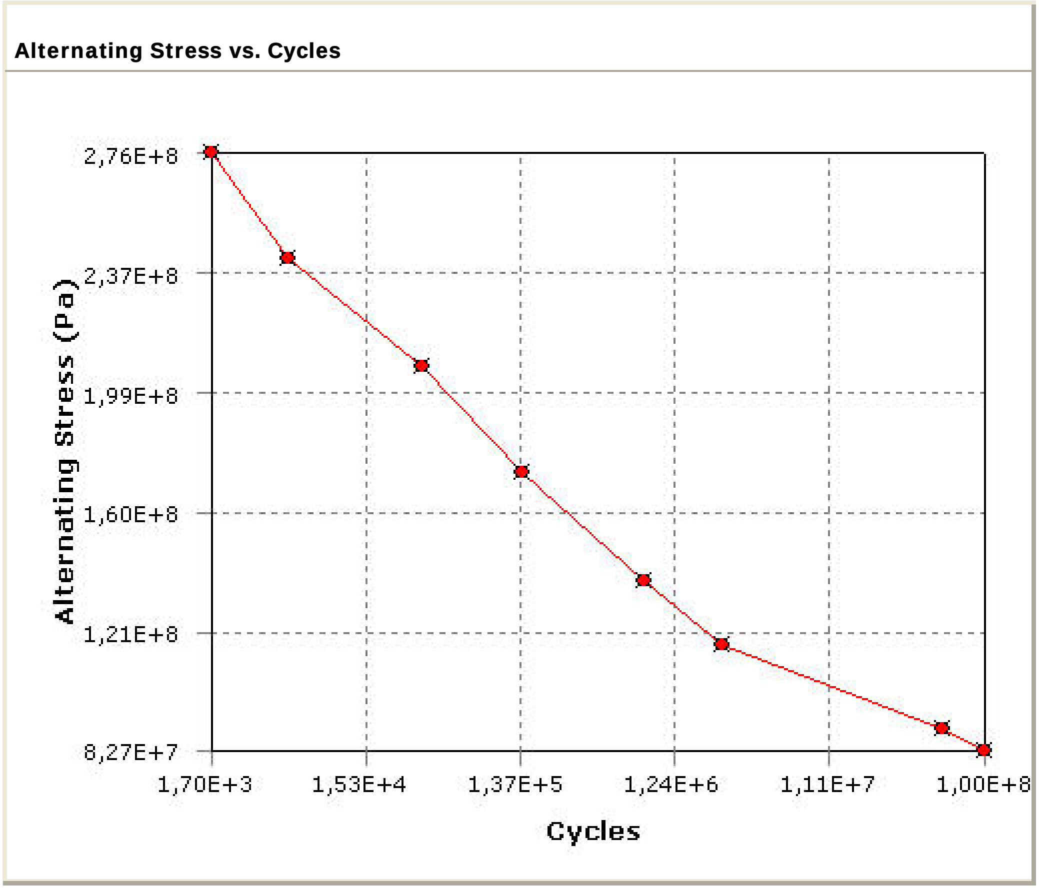
Name	**Type**	**Value**	**Temperature**
Modulus of Elasticity	Temperature-Independent	7.1×10^{10} Pa	
Poisson's Ratio	Temperature-Independent	0.33	
Mass Density	Temperature-Independent	2,770.0 kg/m³	
Coefficient of Thermal Expansion	Temperature-Independent	2.3×10^{-5} 1/°C	
Thermal Conductivity	Temperature-Dependent	114.0 W/m·°C	-100.0 °C
Thermal Conductivity	Temperature-Dependent	144.0 W/m·°C	0.0 °C
Thermal Conductivity	Temperature-Dependent	165.0 W/m·°C	100.0 °C
Thermal Conductivity	Temperature-Dependent	175.0 W/m·°C	200.0 °C
Specific Heat	Temperature-Independent	875.0 J/kg·°C	

***"Aluminum Alloy"* Stress Limits**

Name	**Type**	**Value**
Tensile Yield Strength	Temperature-Independent	2.8×10^{8} Pa
Tensile Ultimate Strength	Temperature-Independent	3.1×10^{8} Pa
Compressive Yield Strength	Temperature-Independent	2.8×10^{8} Pa
Compressive Ultimate Strength	Temperature-Independent	0.0 Pa

- Description: *"General aluminum alloy. Fatigue properties come from MIL-HDBK-5H, page 3-277."*
- *"Aluminum Alloy"* contains nonlinear data for thermal conductivity. Thermal results for bodies using this material usually require several iterations to converge.
- Material data file: *"C:\Program Files\Ansys Inc\v81\AISOL\CommonFiles\Language\en-us\EngineeringData\Materials\Aluminum_Alloy.xml"*


Thermal Conductivity vs. Temperature
175,000
162,800
150,600
138,400
126,200
114,000
Conductivity (W/m·°C)
-100,000
-40,000
20,000
80,000
140,000
200,000
Temperature °C


Alternating Stress vs. Cycles
Alternating Stress (Pa)
2,76E+8
2,37E+8
1,99E+8
1,60E+8
1,21E+8
8,27E+7
1,70E+3
1,53E+4
1,37E+5
1,24E+6
1,11E+7
1,00E+8
Cycles

D.2 Model 2

Author

OSCAR FERNANDEZ

Subject

Horizontal Analysis of the final NEG Coating Lifting Assembly

Prepared For

NEG Coating for TAN Vacuum Chamber

Project Created

Wednesday, December 01, 2004 at 3:48:09 PM

Project Last Modified

Wednesday, May 25, 2005 at 4:59:20 PM

Report Created

Thursday, May 26, 2005 at 3:57:45 PM

Software Used

ANSYS 8.1

Database

G:\Divisions\EST\Groups\IC\LC\Oscar\DOCUMENTOS\PROYECTO\TAN\Pruebas vacio\ansys workbench\HORIZONTAL_4.dsdb

D.2.1 Summary

This report documents design and analysis information created and maintained using the ANSYS® engineering software program. Each scenario listed below represents one complete engineering simulation.

Scenario 1

- Based on the Autodesk® Mechanical Desktop® assembly *"G:\Divisions\EST\Groups\IC\LC\Oscar\DOCUMENTOS\PROYECTO\TAN\Pruebas vacio\ansys workbench\HORIZONTAL_1.dwg"*.
- Considered the effect of body-to-body contact, acceleration, structural loads and structural supports.
- Calculated structural results.
- No alert criteria defined.
- See Scenario 1 below for supporting details and Appendix A1 for corresponding figures.

D.2.2 Introduction

The ANSYS CAE (Computer-Aided Engineering) software program was used in conjunction with 3D CAD (Computer-Aided Design) solid geometry to simulate the behavior of mechanical bodies under thermal/structural loading conditions. ANSYS automated FEA (Finite Element Analysis) technologies from ANSYS, Inc. to generate the results listed in this report.

Each scenario presented below represents one complete engineering simulation. The definition of a simulation includes known factors about a design such as material properties per body, contact behavior between bodies (in an assembly), and types and magnitudes of loading conditions. The results of a simulation provide insight into how the bodies may perform and how the design might be improved. Multiple scenarios allow comparison of results given different loading conditions, materials or geometric configurations.

Convergence and alert criteria may be defined for any of the results and can serve as guides for evaluating the quality of calculated results and the acceptability of values in the context of known design requirements.

- *Solution history* provides a means of assessing the quality of results by examining how values change during successive iterations of solution refinement. *Convergence criteria* sets a specific limit on the allowable change in a result between iterations. A result meeting this criteria is said to be "converged".

- *Alert criteria* define "allowable" ranges for result values. Alert ranges typically represent known aspects of the design specification.

The discussions below follow the organization of information in the ANSYS "Explorer" user interface. Each scenario corresponds to a unique branch in the Explorer "Outline". Names emphasized in *"double quotes"* match preferences set in the user interface.

All values are presented in the *"SI Metric (m, kg, N, °C, s, V, A)"* unit system.

Notice

Do not accept or reject a design based solely on the data presented in this report. Evaluate designs by considering this information in conjunction with experimental test data and the practical experience of design engineers and analysts. A quality approach to engineering design usually mandates physical testing as the final means of validating structural integrity to a measured precision.

D.2.3 "Model"

"Model" obtains geometry from the Autodesk® Mechanical Desktop® assembly *"G:\Divisions\EST\Groups\IC\LC\Oscar\DOCUMENTOS\PROYECTO\TAN\Pruebas vacio\ansys workbench\HORIZONTAL_1.dwg"*.

- The bounding box for all positioned bodies in the model measures 0.27 by 0.43 by 8.5 m along the global x, y and z axes, respectively.
- The model has a total mass of 223.22 kg.
- The model has a total volume of 8.06×10^{-2} m^3.

Bodies

Name	Material	Bounding Box (m)	Mass (kg)	Volume (m^3)	Nodes	Elements
"SMALL_BEAM"	*"Aluminum Alloy"*	0.15, 0.15, 2.0	28.99	1.05×10^{-2}	2560	486
"BIG_BEAM"	*"Aluminum Alloy"*	0.1, 0.22, 6.5	71.25	2.57×10^{-2}	14110	7310
"LONG_UP_PLATE"	*"Aluminum Alloy"*	0.2, 0.01, 7.78	43.06	1.55×10^{-2}	5927	2724
"SHORT_UP_PLATE"	*"Aluminum Alloy"*	0.2, 0.01, 0.64	3.55	1.28×10^{-3}	141	16
"COQ_1"	*"Aluminum Alloy"*	0.2, 0.2, 0.03	3.13	1.13×10^{-3}	325	36
"COQ2_1"	*"Aluminum Alloy"*	0.27, 0.2, 0.03	2.79	1.01×10^{-3}	276	29
"BUT_1"	*"Aluminum Alloy"*	0.2, 0.2, 0.06	4.64	1.68×10^{-3}	403	50
"LAT_PLATE_1"	*"Aluminum Alloy"*	1.0×10^{-2}, 0.21, 5.77	32.91	1.19×10^{-2}	200	16
"LAT_PLATE_2"	*"Aluminum Alloy"*	1.0×10^{-2}, 0.21, 5.77	32.91	1.19×10^{-2}	200	16

Body Groupings

Name	Body Names	Bounding Box (m)	Mass (kg)	Volume (m^3)	Nodes	Elements
"UPPER_PLATE"	LONG_UP_PLATE , SHORT_UP_PLATE	200.0, 10.0, 8,500.0	46.61	1.68×10^{7}	6,068.0	2,740.0

D.2.4 Contact

- *"Contact"* uses a tolerance of 0.0 for automatic detection.

Contact Conditions

Name	Type	Associated Bodies	Scope	Normal Stiffness	Scope Mode	Behavior	Formulation	Thermal Conductance	Pinball Region
"Contact Region"	Bonded	*"BIG_BEAM"* and *"SMALL_BEAM"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 2"	Bonded	*"LONG_UP_PLATE"* and *"SMALL_BEAM"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 5"	Bonded	*"LONG_UP_PLATE"* and *"BIG_BEAM"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 6"	Bonded	*"SHORT_UP_PLATE"* and *"BIG_BEAM"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 10"	Bonded	*"LAT_PLATE_1"* and *"BIG_BEAM"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 11"	Bonded	*"LAT_PLATE_2"* and *"BIG_BEAM"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 12"	Bonded	*"COQ_1"* and *"LONG_UP_PLATE"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 13"	Bonded	*"COQ2_1"* and *"LONG_UP_PLATE"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 16"	Bonded	*"BUT_1"* and *"SHORT_UP_PLATE"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled

D.2.5 Mesh

- *"Mesh"*, associated with *"Model"* has a curvature/proximity value of 0.
- *"Mesh"*, is using the default element size.
- *"Mesh"* uses standard shape checking.
- *"Mesh"* uses a program controlled method for selecting high or low order elements for solids.
- *"Mesh"* uses active assembly for initial size seed.
- *"Mesh"* contains 24142 nodes and 10683 elements.

No mesh controls specified.

D.2.6 "Environment"

"Environment" contains all loading conditions defined for *"Model"* in this scenario.

Standard Earth Gravity - Standard Earth Gravity (Figure A1.1)

- Magnitude: 9.81 m/s^2
- Vector: [0.0 m/s^2 x, 9.81 m/s^2 y, 0.0 m/s^2 z] in the global coordinate system

The following tables list local loads and supports applied to specific geometry.

D.2.7 Structural Loading

Structural Loads

Name	Type	Magnitude	Vector	Reaction Force	Reaction Force Vector	Reaction Moment	Reaction Moment Vector	Associated Bodies
"Force"	Surface Force	1,100.0 N	[0.0 N x, -1,100.0 N y,0.0 N z]	N/A	N/A	N/A	N/A	*"COQ2_1"*
"Force 2"	Surface Force	900.0 N	[0.0 N x, -900.0 N y,0.0 N z]	N/A	N/A	N/A	N/A	*"COQ_1"*
"Force 3"	Surface Force	2,000.0 N	[0.0 N x, -2,000.0 N y,0.0 N z]	N/A	N/A	N/A	N/A	*"LONG_UP_PLATE"* and *"SHORT_UP_PLATE"*

D.2.8 Structural Supports

Structural Supports

Name	Type	Reaction Force	Reaction Force Vector	Reaction Moment	Reaction Moment Vector	Associated Bodies
"Fixed Support"	Fixed Edge	24,274.39 N (not updated)	[2.15 N x, 2,415.73 N y, -24,153.89 N z] (not updated)	1.66 N·m (not updated)	[6.7×10^{-13} N·m x, 1.66 N·m y, 0.1 N·m z] (not updated)	*"SMALL_BEAM"*
"Fixed Support 2"	Fixed Edge	24,445.51 N (not updated)	[-2.15 N x, 3,764.67 N y, -24,153.89 N z] (not updated)	3.31 N·m (not updated)	[0.0 N·m x, -2.66 N·m y, -1.97 N·m z] (not updated)	*"BIG_BEAM"*

D.2.9 "Solution"

"Solution" contains the calculated response for *"Model"* given loading conditions defined in *"Environment"*.

It was selected that the program would choose the solver used in this solution.

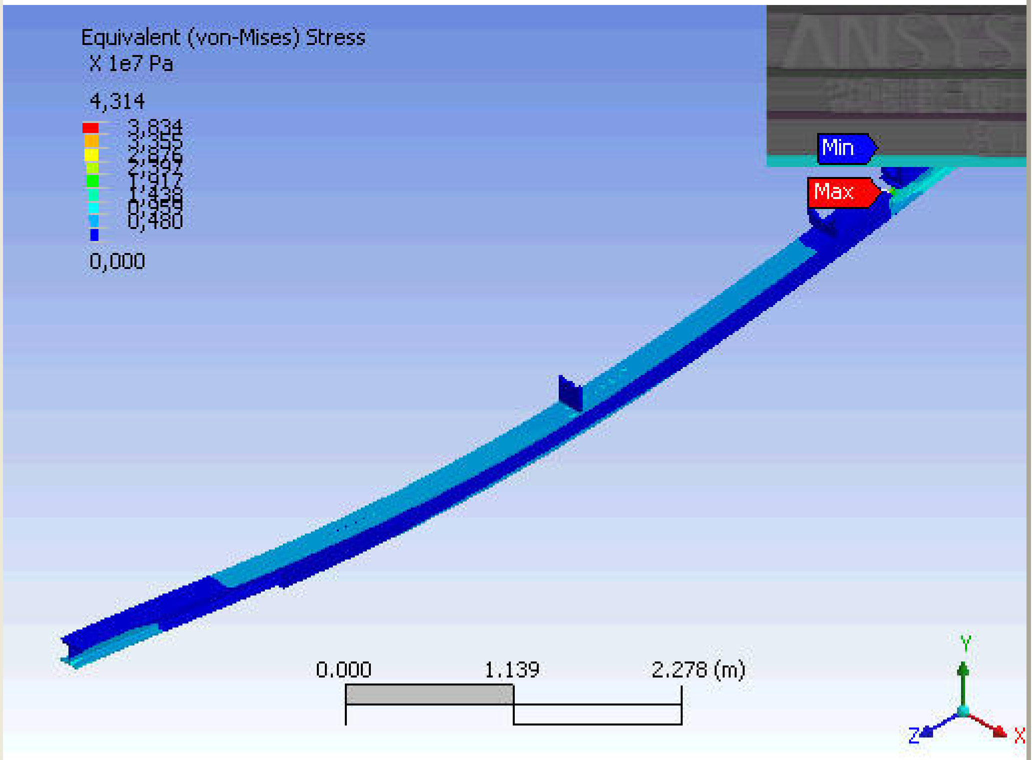
D.2.10 Structural Results

Values

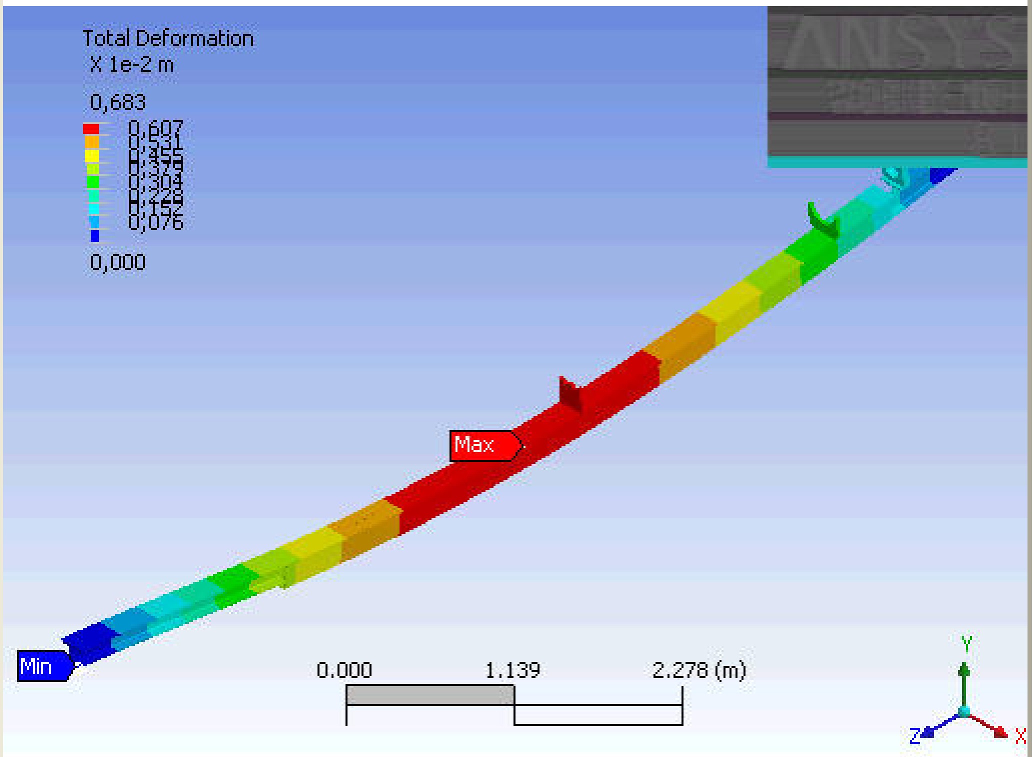
Name	**Figure**	**Scope**	**Minimum**	**Maximum**	**Alert Criteria**
"Equivalent Stress"	A1.2	All Bodies In *"Model"*	3,437.01 Pa	4.31×10^{7} Pa	None
"Total Deformation"	A1.3	All Bodies In *"Model"*	0.0 m	6.83×10^{-3} m	None

D.2.11 Figures

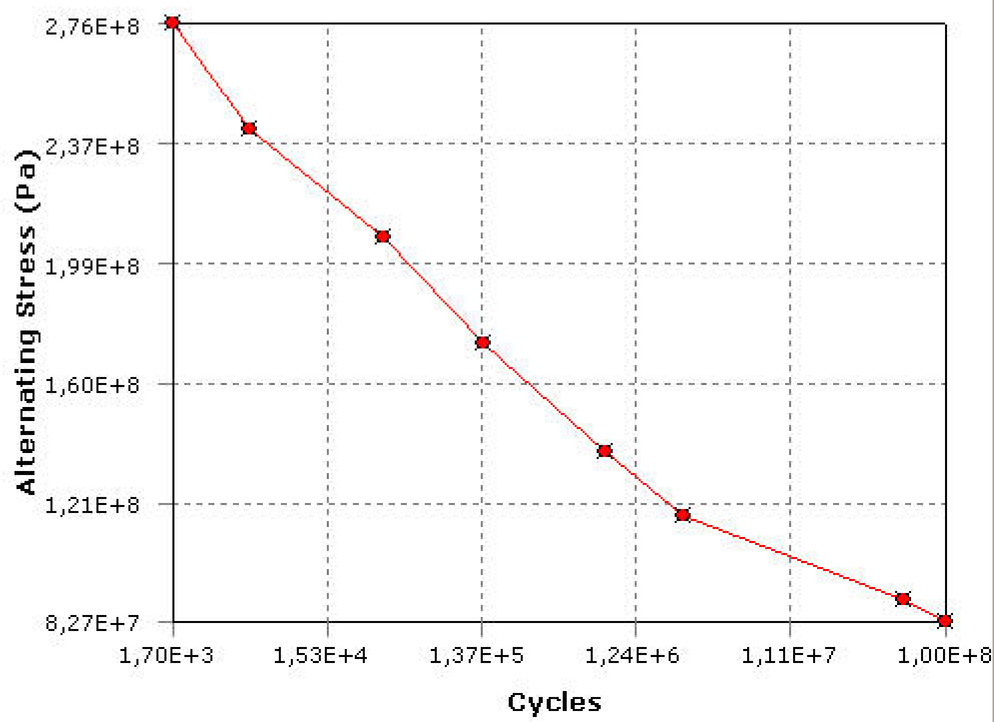
"Equivalent Stress" **Contours**



"Total Deformation" **Contours**



Alternating Stress vs. Cycles



D.3 Model 3

ANSYS® Report

Author

OSCAR FERNANDEZ

Subject

Vertical Analisis of the NEG Coating Lifting Assembly (1 Central Support Plate)

Prepared For

NEG Coating for TAN VAcuum Chamber

Project Created

Monday, November 29, 2004 at 2:31:38 PM

Project Last Modified

Monday, November 29, 2004 at 3:01:47 PM

Report Created

Monday, November 29, 2004 at 7:06:32 PM

Software Used

ANSYS 8.1

Database

G:\Divisions\EST\Groups\IC\LC\Oscar\DOCUMENTOS\PROYECTO\TAN\Pruebas vacio\ansys workbench\VERTICAL_1.dsdb

D.3.1 Summary

This report documents design and analysis information created and maintained using the ANSYS® engineering software program. Each scenario listed below represents one complete engineering simulation.

Scenario 1

- Based on the Autodesk® Mechanical Desktop® assembly *"G:\Divisions\EST\Groups\IC\LC\Oscar\DOCUMENTOS\PROYECTO\TAN\Pruebas vacio\ansys workbench\dosvigas_y_placa_cortada_extremo(nuevos_agujeros)_coquis_but_placas.dwg"*.
- Considered the effect of body-to-body contact, acceleration, structural loads and structural supports.
- Calculated structural results.
- No convergence criteria defined.
- No alert criteria defined.
- See Scenario 1 below for supporting details and Appendix A1 for corresponding figures.

D.3.2 Introduction

The ANSYS CAE (Computer-Aided Engineering) software program was used in conjunction with 3D CAD (Computer-Aided Design) solid geometry to simulate the behavior of mechanical bodies under thermal/structural loading conditions. ANSYS automated FEA (Finite Element Analysis) technologies from ANSYS, Inc. to generate the results listed in this report.

Each scenario presented below represents one complete engineering simulation. The definition of a simulation includes known factors about a design such as material properties per body, contact behavior between bodies (in an assembly), and types and magnitudes of loading conditions. The results of a simulation provide insight into how the bodies may perform and how the design might be improved. Multiple scenarios allow comparison of results given different loading conditions, materials or geometric configurations.

Convergence and alert criteria may be defined for any of the results and can serve as guides for evaluating the quality of calculated results and the acceptability of values in the context of known design requirements.

- *Solution history* provides a means of assessing the quality of results by examining how values change during successive iterations of solution refinement. *Convergence criteria* sets a specific limit on the allowable change in a result between iterations. A result meeting this criteria is said to be "converged".

- *Alert criteria* define "allowable" ranges for result values. Alert ranges typically represent known aspects of the design specification.

The discussions below follow the organization of information in the ANSYS "Explorer" user interface. Each scenario corresponds to a unique branch in the Explorer "Outline". Names emphasized in *"double quotes"* match preferences set in the user interface.

All values are presented in the *"SI Metric (m, kg, N, °C, s, V, A)"* unit system.

Notice

Do not accept or reject a design based solely on the data presented in this report. Evaluate designs by considering this information in conjunction with experimental test data and the practical experience of design engineers and analysts. A quality approach to engineering design usually mandates physical testing as the final means of validating structural integrity to a measured precision.

D.3.3 "Model"

"Model" obtains geometry from the Autodesk® Mechanical Desktop® assembly *"G:\Divisions\EST\Groups\IC\LC\Oscar\DOCUMENTOS\PROYECTO\TAN\Pruebas vacio\ansys workbench\dosvigas_y_placa_cortada_extremo(nuevos_agujeros)_coquis_but_placas.dwg"*.

- The bounding box for all positioned bodies in the model measures 0.27 by 0.43 by 8.5 m along the global x, y and z axes, respectively.
- The model has a total mass of 171.64 kg.
- The model has a total volume of 6.2×10^{-2} m³.

Bodies

Name	**Material**	**Bounding Box (m)**	**Mass (kg)**	**Volume (m³)**	**Nodes**	**Elements**
"SHORT_BEAM"	*"Aluminum Alloy"*	0.15, 0.15, 2.0	28.99	1.05×10^{-2}	2823	540
"LONG_BEAM"	*"Aluminum Alloy"*	0.1, 0.22, 6.5	71.25	2.57×10^{-2}	14582	7584
"L_UP_PLATE"	*"Aluminum Alloy"*	0.2, 0.01, 7.78	43.06	1.55×10^{-2}	10946	1918
"S_UP_PLATE"	*"Aluminum Alloy"*	0.2, 0.01, 0.64	3.55	1.28×10^{-3}	141	16
"COQ_1"	*"Aluminum Alloy"*	0.2, 0.2, 0.03	3.13	1.13×10^{-3}	325	36
"COQ2_1"	*"Aluminum Alloy"*	0.27, 0.2, 0.03	2.79	1.01×10^{-3}	276	29
"BUT_1"	*"Aluminum Alloy"*	0.2, 0.2, 0.06	4.64	1.68×10^{-3}	403	50
"PLATE_1"	*"Aluminum Alloy"*	8.0×10^{-3}, 0.22, 1.0	4.78	1.73×10^{-3}	165	16
"PLATE_2"	*"Aluminum Alloy"*	8.0×10^{-3}, 0.22, 1.0	4.78	1.73×10^{-3}	165	16
"HOLE_PLATE_1"	*"Aluminum Alloy"*	1.5×10^{-2}, 0.16, 0.36	2.34	8.44×10^{-4}	337	37
"HOLE_PLATE_2"	*"Aluminum Alloy"*	1.5×10^{-2}, 0.16, 0.36	2.34	8.44×10^{-4}	337	37

TBody Groupings

Name	**Body Names**	**Bounding Box (m)**	**Mass (kg)**	**Volume (m³)**	**Nodes**	**Elements**
"UPPER_PLATE"	L_UP_PLATE , S_UP_PLATE	200.0, 10.0, 8,500.0	46.61	1.68×10^{7}	11,087.0	1,934.0

D.3.4 Contact

- *"Contact"* uses a tolerance of 0.0 for automatic detection.

Table 3.1.1.1. Contact Conditions

Name	Type	Associated Bodies	Scope	Normal Stiffness	Scope Mode	Behavior	Formulation	Thermal Conductance	Pinball Region
"Contact Region"	Bonded	*"LONG_BEAM"* and *"SHORT_BEAM"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 2"	Bonded	*"L_UP_PLATE"* and *"SHORT_BEAM"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 3"	Bonded	*"L_UP_PLATE"* and *"LONG_BEAM"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 4"	Bonded	*"S_UP_PLATE"* and *"LONG_BEAM"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 6"	Bonded	*"COQ2_1"* and *"LONG_BEAM"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 8"	Bonded	*"PLATE_1"* and *"LONG_BEAM"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 9"	Bonded	*"PLATE_2"* and *"LONG_BEAM"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 10"	Bonded	*"HOLE_PLATE_1"* and *"LONG_BEAM"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 11"	Bonded	*"HOLE_PLATE_2"* and *"LONG_BEAM"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 12"	Bonded	*"COQ_1"* and *"L_UP_PLATE"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 13"	Bonded	*"COQ2_1"* and *"L_UP_PLATE"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 16"	Bonded	*"BUT_1"* and *"S_UP_PLATE"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled

D.3.5 Mesh

- *"Mesh"*(Figure A1.1) , associated with *"Model"* has an overall relevance of 0.
- *"Mesh"* contains 30500 nodes and 10279 elements.

No mesh controls specified.

D.3.6 "Environment"

"Environment"(Figure A1.2) contains all loading conditions defined for *"Model"* in this scenario.

Standard Earth Gravity - Standard Earth Gravity

- Magnitude: 9.81 m/s²
- Vector: [0.0 m/s² x, 0.0 m/s² y, 9.81 m/s² z] in the global coordinate system

The following tables list local loads and supports applied to specific geometry.

D.3.7 Structural Loading

Structural Loads

Name	Type	Magnitude	Vector	Reaction Force	Reaction Force Vector	Reaction Moment	Reaction Moment Vector	Associated Bodies
"Force"	Surface Force	3,000.0 N	[3.67×10^{-13} N x, 4.08×10^{-29} N y,-3,000.0 N z]	N/A	N/A	N/A	N/A	*"BUT_1"*

D.3.8 Structural Supports

Structural Supports

Name	Type	Reaction Force	Reaction Force Vector	Reaction Moment	Reaction Moment Vector	Associated Bodies
"Fixed Support"	Fixed Surface	3,720.78 N	[-5.64×10^{-2} N x, 18.92 N y, 3,720.74 N z]	584.63 N·m	[584.6 N·m x, 6.22 N·m y, -3.8×10^{-2} N·m z]	*"LONG_BEAM"*
"Fixed Support 2"	Fixed Edge	951.62 N	[5.64×10^{-2} N x, -19.02 N y, 951.43 N z]	25.76 N·m	[-25.76 N·m x, -0.22 N·m y, 5.64×10^{-3} N·m z]	*"SHORT_BEAM"*

D.3.9 "Solution"

"Solution" contains the calculated response for *"Model"* given loading conditions defined in *"Environment"*.

It was selected that the program would choose the solver used in this solution.

D.3.10 Structural Results

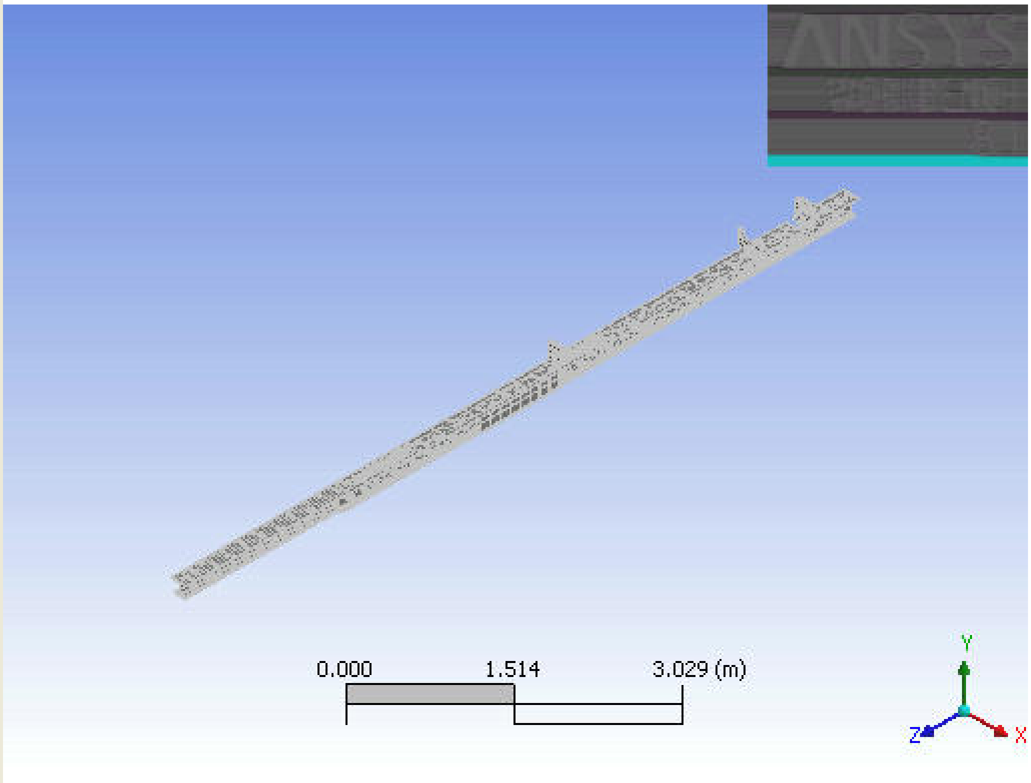
Table 3.3.1.1. Values

Name	Figure	Scope	Minimum	Maximum	Alert Criteria
"Equivalent Elastic Strain"	A1.3	All Bodies In *"Model"*	2.21×10^{-9} m/m	1.46×10^{-4} m/m	None
"Total Deformation"	A1.4	All Bodies In *"Model"*	0.0 m	1.03×10^{-4} m	None

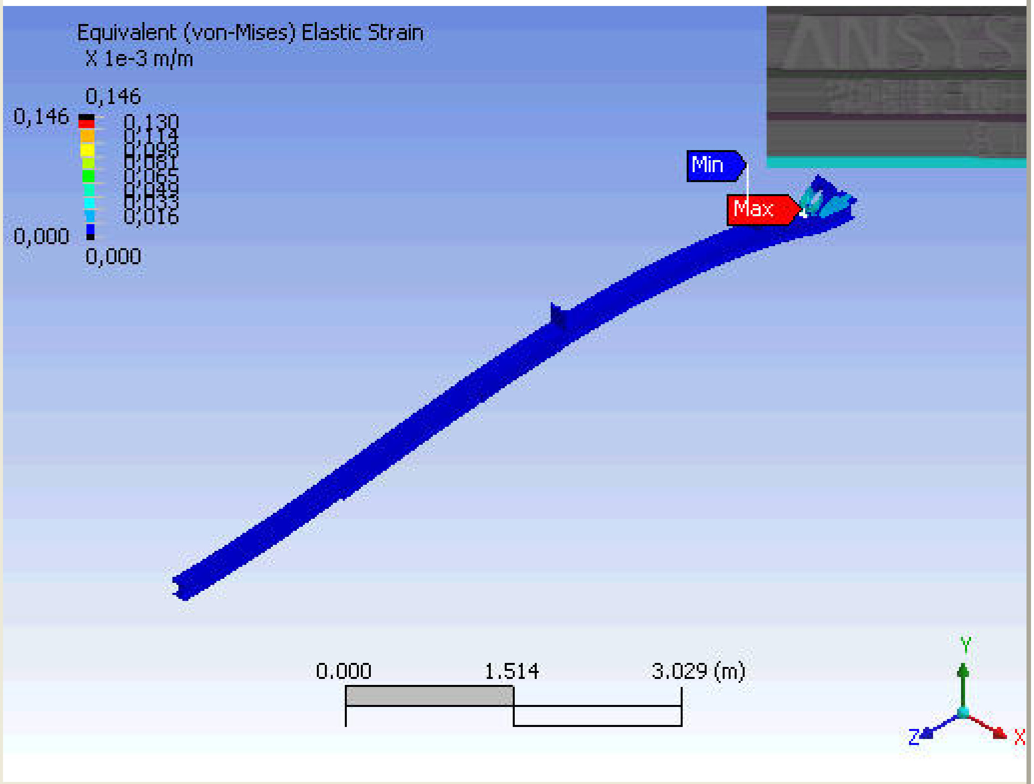
- Convergence tracking not enabled.

D.3.11 Figures

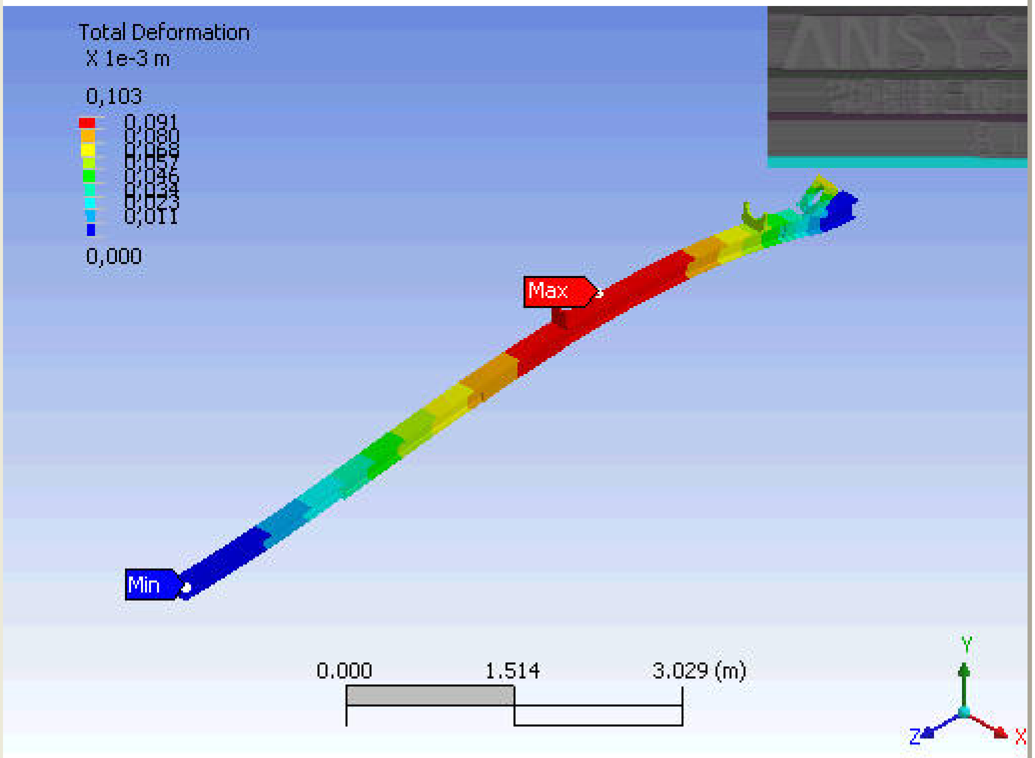
***"Mesh"* Geometry**

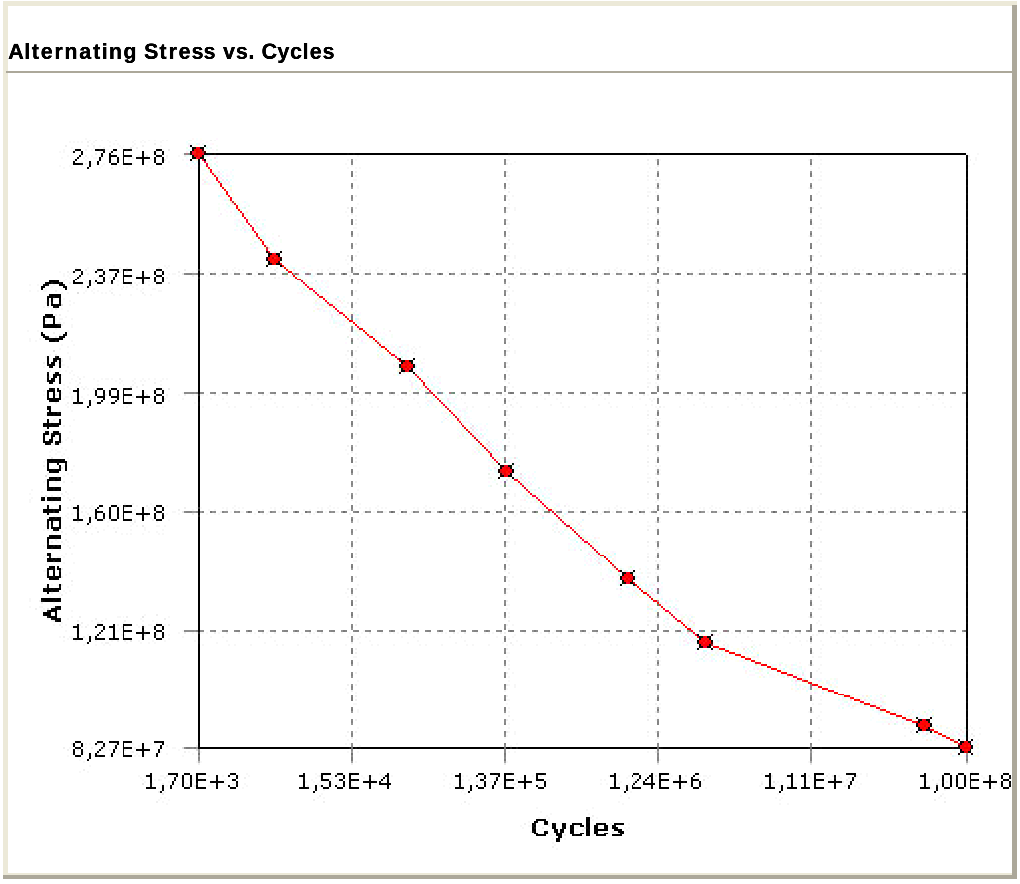


***"Equivalent Elastic Strain"* Contours**



"Total Deformation" **Contours**




Alternating Stress vs. Cycles
Alternating Stress (Pa)
2,76E+8
2,37E+8
1,99E+8
1,60E+8
1,21E+8
8,27E+7
1,70E+3
1,53E+4
1,37E+5
1,24E+6
1,11E+7
1,00E+8
Cycles

D.4 Model 4

ANSYS Report

Author

OSCAR FERNANDEZ

Subject

Horizontal Analysis of the two beams + local cut (F1 200 Kg)

Prepared For

NEG Coating for TAN Vacuum Chamber

Project Created

Tuesday, November 23, 2004 at 4:53:04 PM

Project Last Modified

Tuesday, November 23, 2004 at 6:56:41 PM

Report Created

Thursday, May 26, 2005 at 3:28:22 PM

Software Used

ANSYS 8.1

Database

G:\Divisions\EST\Groups\IC\LC\Oscar\DOCUMENTOS\PROYECTO\TAN\Pruebas vacio\ansys workbench\dosvigas_y_placa_cortada_extremo(nuevos_agujeros)_repartida200Kg.dsdb

D.4.1 Summary

This report documents design and analysis information created and maintained using the ANSYS® engineering software program. Each scenario listed below represents one complete engineering simulation.

Scenario 1

- Based on the Autodesk® Mechanical Desktop® assembly *"G:\Divisions\EST\Groups\IC\LC\Oscar\DOCUMENTOS\PROYECTO\TAN\Pruebas vacio\ansys workbench\dosvigas_y_placa_cortada_extremo(nuevos_agujeros).dwg"*.
- Considered the effect of body-to-body contact, acceleration, structural loads and structural supports.
- Calculated structural results.
- No convergence criteria defined.
- No alert criteria defined.
- See Scenario 1 below for supporting details and Appendix A1 for corresponding figures.

D.4.2 Introduction

The ANSYS CAE (Computer-Aided Engineering) software program was used in conjunction with 3D CAD (Computer-Aided Design) solid geometry to simulate the behavior of mechanical bodies under thermal/structural loading conditions. ANSYS automated FEA (Finite Element Analysis) technologies from ANSYS, Inc. to generate the results listed in this report.

Each scenario presented below represents one complete engineering simulation. The definition of a simulation includes known factors about a design such as material properties per body, contact behavior between bodies (in an assembly), and types and magnitudes of loading conditions. The results of a simulation provide insight into how the bodies may perform and how the design might be improved. Multiple scenarios allow comparison of results given different loading conditions, materials or geometric configurations.

Convergence and alert criteria may be defined for any of the results and can serve as guides for evaluating the quality of calculated results and the acceptability of values in the context of known design requirements.

- *Solution history* provides a means of assessing the quality of results by examining how values change during successive iterations of solution refinement. *Convergence criteria* sets a specific limit on the allowable change in a result between iterations. A result meeting this criteria is said to be "converged".

- *Alert criteria* define "allowable" ranges for result values. Alert ranges typically represent known aspects of the design specification.

The discussions below follow the organization of information in the ANSYS "Explorer" user interface. Each scenario corresponds to a unique branch in the Explorer "Outline". Names emphasized in *"double quotes"* match preferences set in the user interface.

All values are presented in the *"SI Metric (m, kg, N, °C, s, V, A)"* unit system.

Notice

Do not accept or reject a design based solely on the data presented in this report. Evaluate designs by considering this information in conjunction with experimental test data and the practical experience of design engineers and analysts. A quality approach to engineering design usually mandates physical testing as the final means of validating structural integrity to a measured precision.

D.4.3 "Model"

"Model" obtains geometry from the Autodesk® Mechanical Desktop® assembly *"G:\Divisions\EST\Groups\IC\LC\Oscar\DOCUMENTOS\PROYECTO\TAN\Pruebas vacio\ansys workbench\dosvigas_y_placa_cortada_extremo(nuevos_agujeros).dwg"*.

- The bounding box for all positioned bodies in the model measures 0.2 by 0.23 by 8.5 m along the global x, y and z axes, respectively.
- The model has a total mass of 146.84 kg.
- The model has a total volume of 5.3×10^{-2} m³.

Bodies

Name	Material	Bounding Box (m)	Mass (kg)	Volume (m³)	Nodes	Elements
"PART1_1"	*"Aluminum Alloy"*	0.15, 0.15, 2.0	28.99	1.05×10^{-2}	2823	540
"PART2_1"	*"Aluminum Alloy"*	0.1, 0.22, 6.5	71.25	2.57×10^{-2}	14779	7734
"Part"	*"Aluminum Alloy"*	0.2, 0.01, 7.78	43.06	1.55×10^{-2}	10946	1918
"Part 2"	*"Aluminum Alloy"*	0.2, 0.01, 0.64	3.55	1.28×10^{-3}	141	16

Body Groupings

Name	Body Names	Bounding Box (m)	Mass (kg)	Volume (m³)	Nodes	Elements
"PART3_1"	Part , Part 2	200.0, 10.0, 8,500.0	46.61	1.68×10^{7}	11,087.0	1,934.0

D.4.4 Contact

- *"Contact"* uses a tolerance of 0.0 for automatic detection.

Contact Conditions

Name	Type	Associated Bodies	Scope	Normal Stiffness	Scope Mode	Behavior	Formulation	Thermal Conductance	Pinball Region
"Contact Region"	Bonded	*"PART2_1"* and *"PART1_1"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 2"	Bonded	*"Part"* and *"PART1_1"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 3"	Bonded	*"Part"* and *"PART2_1"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 4"	Bonded	*"Part 2"* and *"PART2_1"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled

D.4.5 Mesh

- *"Mesh"*, associated with *"Model"* has an overall relevance of 0.
- *"Mesh"* contains 28689 nodes and 10208 elements.

No mesh controls specified.

D.4.6 "Environment"

"Environment"(Figure A1.1) contains all loading conditions defined for *"Model"* in this scenario.

Standard Earth Gravity - Standard Earth Gravity

- Magnitude: 9.81 m/s^2
- Vector: [0.0 m/s^2 x, 9.81 m/s^2 y, 0.0 m/s^2 z] in the global coordinate system

The following tables list local loads and supports applied to specific geometry.

D.4.7 Structural Loading

Structural Loads

Name	Type	Magnitude	Vector	Reaction Force	Reaction Force Vector	Reaction Moment	Reaction Moment Vector	Associated Bodies
"Force"	Surface Force	2,000.0 N	[0.0 N x, -2,000.0 N y,0.0 N z]	N/A	N/A	N/A	N/A	*"Part"* and *"Part 2"*

D.4.8 Structural Supports

Structural Supports

Name	Type	Reaction Force	Reaction Force Vector	Reaction Moment	Reaction Moment Vector	Associated Bodies
"Fixed Support"	Fixed Edge	13,368.11 N	[-0.41 N x, 1,778.62 N y, 13,249.26 N z]	20.93 N·m	[0.0 N·m x, 20.72 N·m y, -2.99 N·m z]	*"PART2_1"*
"Fixed Support 2"	Fixed Edge	13,351.62 N	[0.41 N x, 1,650.12 N y, -13,249.26 N z]	0.67 N·m	[3.68×10^{-13} N·m x, -0.66 N·m y, 0.1 N·m z]	*"PART1_1"*

D.4.9 "Solution"

"Solution" contains the calculated response for *"Model"* given loading conditions defined in *"Environment"*.

It was selected that the program would choose the solver used in this solution.

D.4.10 Structural Results

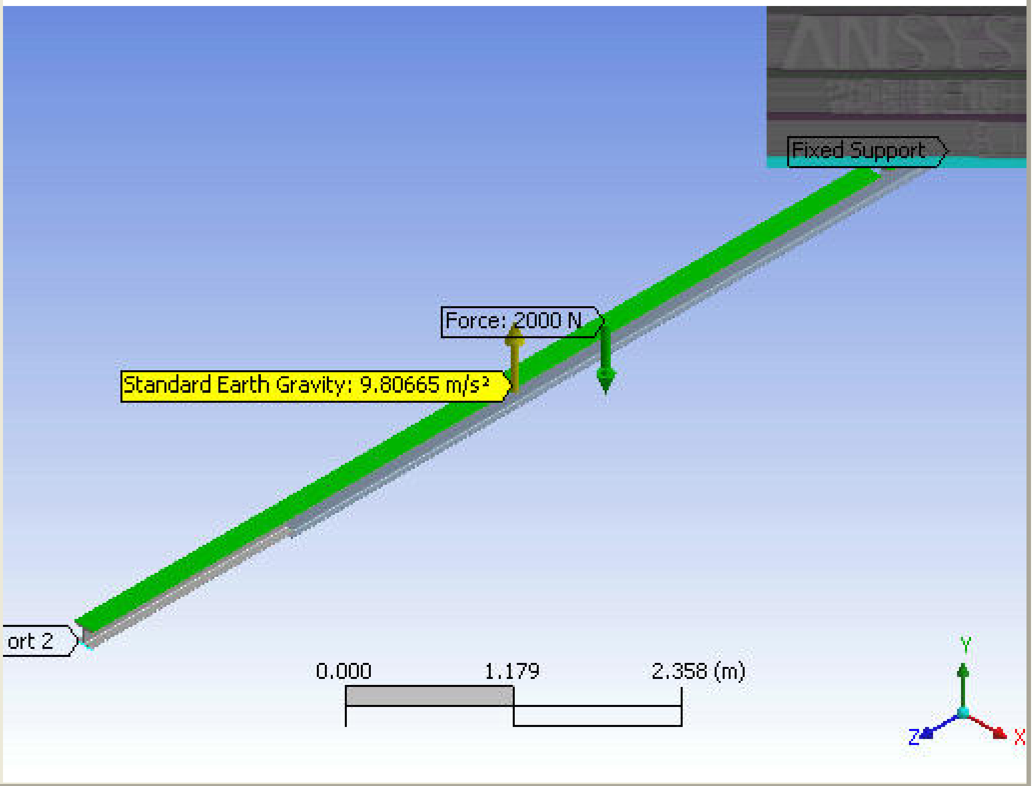
Values

Name	**Figure**	**Scope**	**Minimum**	**Maximum**	**Alert Criteria**
"Equivalent Stress"	A1.2	Surface(s) on *"Part 2"* and *"Part"*	66,299.46 Pa	1.29×10^{7} Pa	None
"Total Deformation"	A1.3	All Bodies In *"Model"*	0.0 m	4.4×10^{-3} m	None

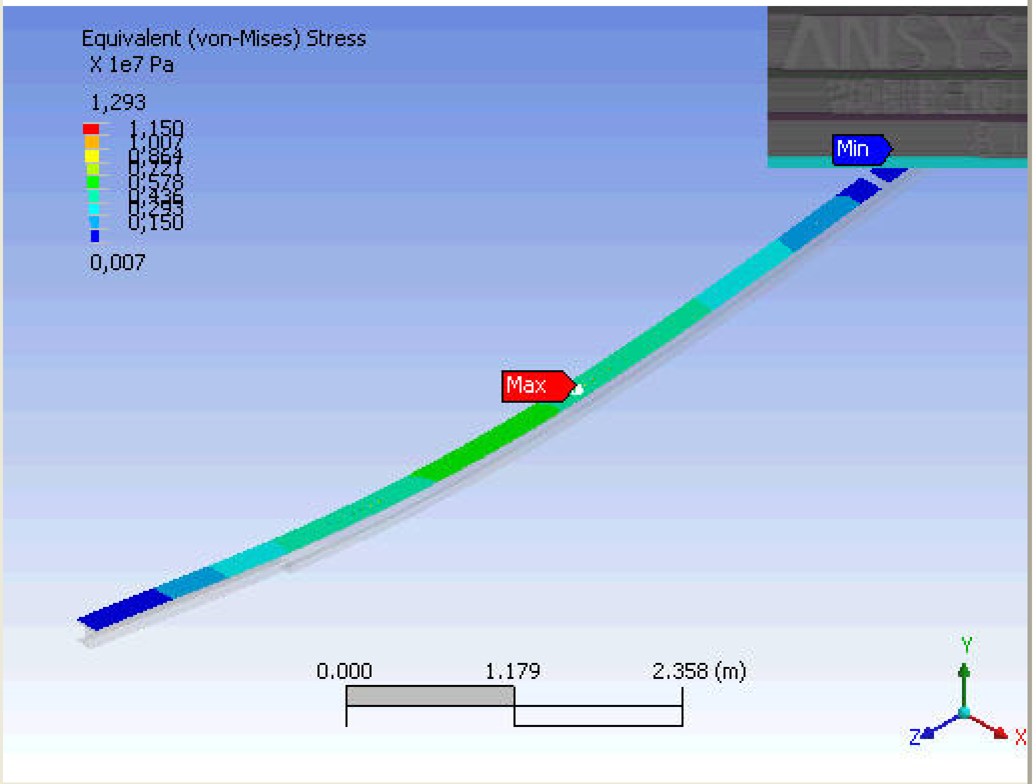
- Convergence tracking not enabled.

D.4.11 Scenario 1 Figures

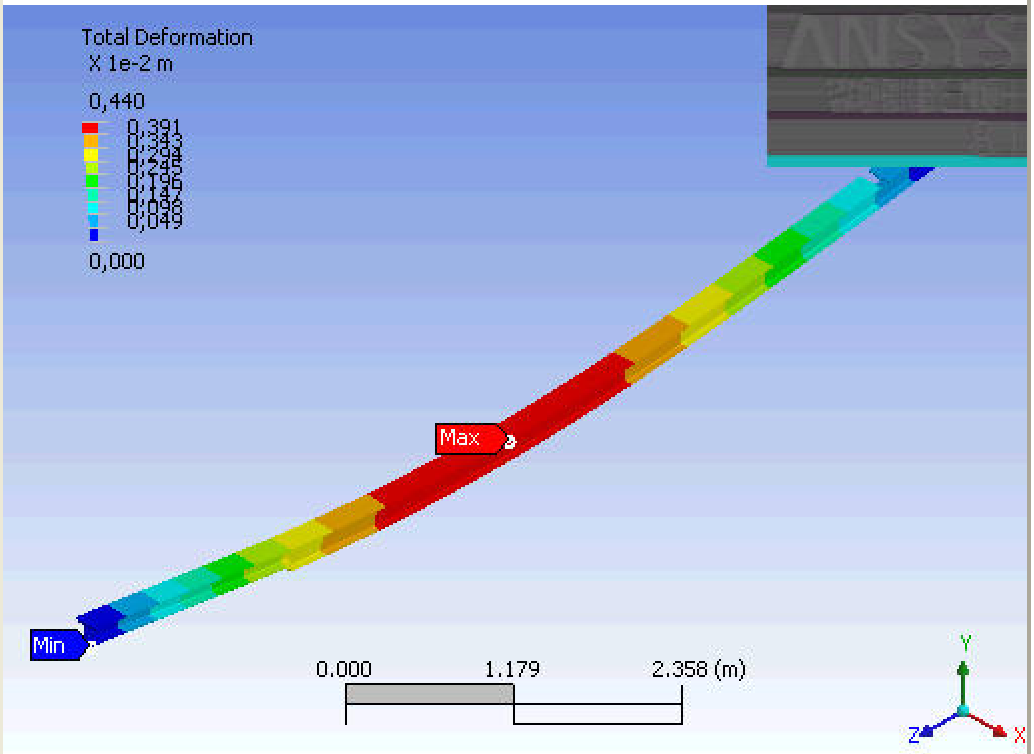
"Environment" **Geometry**

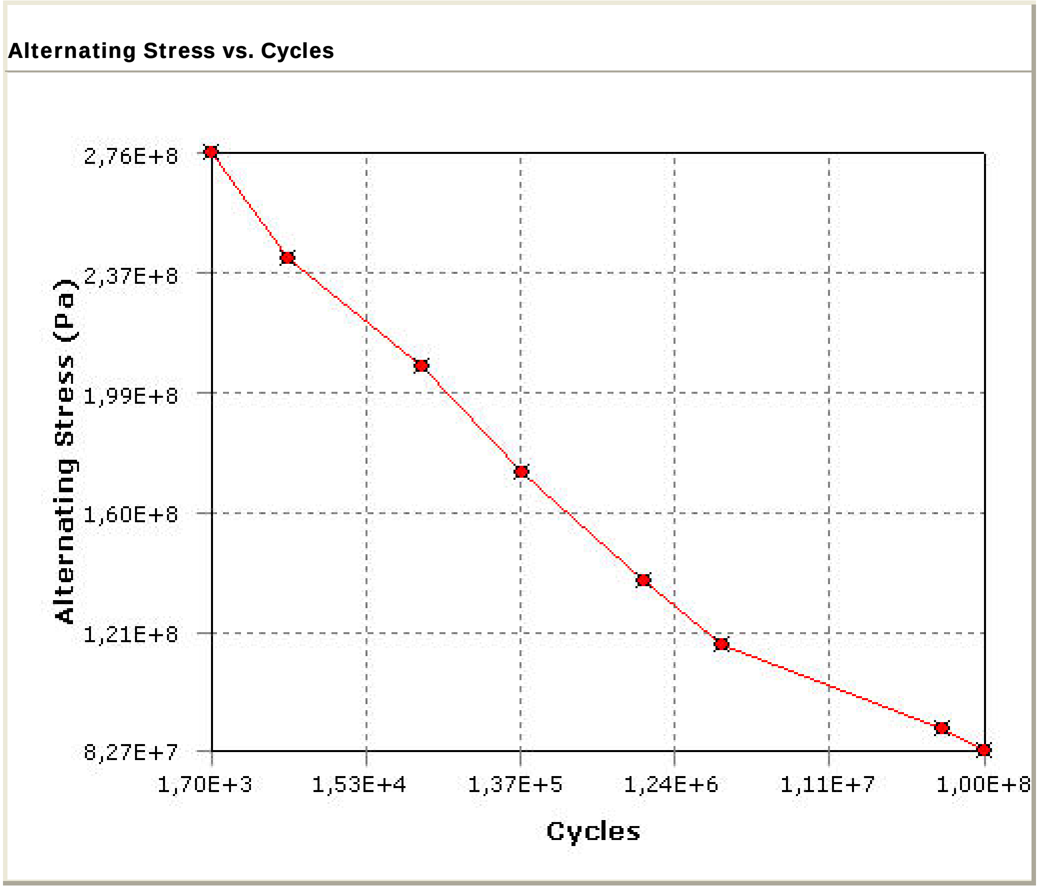


***"Equivalent Stress"* Contours**



***"Total Deformation"* Contours**




Alternating Stress vs. Cycles
2,76E+8
2,37E+8
1,99E+8
1,60E+8
1,21E+8
8,27E+7
Alternating Stress (Pa)
1,70E+3
1,53E+4
1,37E+5
1,24E+6
1,11E+7
1,00E+8
Cycles

D.5 Model 5

Author

OSCAR FERNANDEZ

Subject

Horizontal Analysis (Old Support Situation)

Prepared For

NEG Coating for TAN Vacuum Chamber

Project Created

Tuesday, November 23, 2004 at 4:53:04 PM

Project Last Modified

Thursday, November 25, 2004 at 9:41:25 AM

Report Created

Monday, May 30, 2005 at 1:51:07 PM

Software Used

ANSYS 8.1

Database

G:\Divisions\EST\Groups\IC\LC\Oscar\DOCUMENTOS\PROYECTO\TAN\Pruebas vacio\ansys workbench\dosvigas_y_placa_cortada_extremo(nuevos_agujeros)_repartida200kg_puntuales_anteriores.dsdb

D.5.1 Summary

This report documents design and analysis information created and maintained using the ANSYS® engineering software program. Each scenario listed below represents one complete engineering simulation.

Scenario 1

- Based on the Autodesk® Mechanical Desktop® assembly *"G:\Divisions\EST\Groups\IC\LC\Oscar\DOCUMENTOS\PROYECTO\TAN\Pruebas vacio\ansys workbench\dosvigas_y_placa_cortada_extremo(nuevos_agujeros).dwg"*.
- Considered the effect of body-to-body contact, acceleration, structural loads and structural supports.
- Calculated structural results.
- No convergence criteria defined.
- No alert criteria defined.
- See Scenario 1 below for supporting details and Appendix A1 for corresponding figures.

D.5.2 Introduction

The ANSYS CAE (Computer-Aided Engineering) software program was used in conjunction with 3D CAD (Computer-Aided Design) solid geometry to simulate the behavior of mechanical bodies under thermal/structural loading conditions. ANSYS automated FEA (Finite Element Analysis) technologies from ANSYS, Inc. to generate the results listed in this report.

Each scenario presented below represents one complete engineering simulation. The definition of a simulation includes known factors about a design such as material properties per body, contact behavior between bodies (in an assembly), and types and magnitudes of loading conditions. The results of a simulation provide insight into how the bodies may perform and how the design might be improved. Multiple scenarios allow comparison of results given different loading conditions, materials or geometric configurations.

Convergence and alert criteria may be defined for any of the results and can serve as guides for evaluating the quality of calculated results and the acceptability of values in the context of known design requirements.

- *Solution history* provides a means of assessing the quality of results by examining how values change during successive iterations of solution refinement. *Convergence criteria* sets a specific limit on the allowable change in a result between iterations. A result meeting this criteria is said to be "converged".
- *Alert criteria* define "allowable" ranges for result values. Alert ranges typically represent known aspects of the design specification.

The discussions below follow the organization of information in the ANSYS "Explorer" user interface. Each scenario corresponds to a unique branch in the Explorer "Outline". Names emphasized in *"double quotes"* match preferences set in the user interface.

All values are presented in the *"SI Metric (m, kg, N, °C, s, V, A)"* unit system.

Notice

Do not accept or reject a design based solely on the data presented in this report. Evaluate designs by considering this information in conjunction with experimental test data and the practical experience of design engineers and analysts. A quality approach to engineering design usually mandates physical testing as the final means of validating structural integrity to a measured precision.

D.5.3 "Model"

"Model" obtains geometry from the Autodesk® Mechanical Desktop® assembly *"G:\Divisions\EST\Groups\IC\LC\Oscar\DOCUMENTOS\PROYECTO\TAN\Pruebas vacio\ansys workbench\dosvigas_y_placa_cortada_extremo(nuevos_agujeros).dwg"*.

- The bounding box for all positioned bodies in the model measures 0.2 by 0.23 by 8.5 m along the global x, y and z axes, respectively.
- The model has a total mass of 146.84 kg.
- The model has a total volume of 5.3×10^{-2} m³.

Bodies

Name	Material	Bounding Box (m)	Mass (kg)	Volume (m³)	Nodes	Elements
"PART1_1"	*"Aluminum Alloy"*	0.15, 0.15, 2.0	28.99	1.05×10^{-2}	2823	540
"PART2_1"	*"Aluminum Alloy"*	0.1, 0.22, 6.5	71.25	2.57×10^{-2}	14779	7734
"Part"	*"Aluminum Alloy"*	0.2, 0.01, 7.78	43.06	1.55×10^{-2}	10946	1918
"Part 2"	*"Aluminum Alloy"*	0.2, 0.01, 0.64	3.55	1.28×10^{-3}	141	16

Body Groupings

Name	Body Names	Bounding Box (m)	Mass (kg)	Volume (m³)	Nodes	Elements
"PART3_1"	Part , Part 2	200.0, 10.0, 8,500.0	46.61	1.68×10^{7}	11,087.0	1,934.0

D.5.4 Contact

- *"Contact"* uses a tolerance of 0.0 for automatic detection.

Contact Conditions

Name	Type	Associated Bodies	Scope	Normal Stiffness	Scope Mode	Behavior	Formulation	Thermal Conductance	Pinball Region
"Contact Region"	Bonded	*"PART2_1"* and *"PART1_1"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 2"	Bonded	*"Part"* and *"PART1_1"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 3"	Bonded	*"Part"* and *"PART2_1"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled
"Contact Region 4"	Bonded	*"Part 2"* and *"PART2_1"*	Face, Face	Program Controlled	Automatic	Symmetric	Pure Penalty	Program Controlled	Program Controlled

D.5.5 Mesh

- *"Mesh"*, associated with *"Model"* has an overall relevance of 0.
- *"Mesh"* contains 28689 nodes and 10208 elements.

No mesh controls specified.

D.5.6 "Environment"

"Environment"(Figure A1.1) contains all loading conditions defined for *"Model"* in this scenario.

Standard Earth Gravity - Standard Earth Gravity

- Magnitude: 9.81 m/s²
- Vector: [0.0 m/s² x, 9.81 m/s² y, 0.0 m/s² z] in the global coordinate system

The following tables list local loads and supports applied to specific geometry.

D.5.7 Structural Loading

Structural Loads

Name	**Type**	**Magnitude**	**Vector**	**Reaction Force**	**Reaction Force Vector**	**Reaction Moment**	**Reaction Moment Vector**	**Associated Bodies**
"Force 3"	Surface Force	2,000.0 N	[0.0 N x, -2,000.0 N y,0.0 N z]	N/A	N/A	N/A	N/A	*"Part"* and *"Part 2"*
"Force"	Edge Force	1,100.0 N	[0.0 N x, -1,100.0 N y,0.0 N z]	N/A	N/A	N/A	N/A	*"Part"*
"Force 2"	Edge Force	900.0 N	[0.0 N x, -900.0 N y,0.0 N z]	N/A	N/A	N/A	N/A	*"Part"*

D.5.8 Structural Supports

Structural Supports

Name	**Type**	**Reaction Force**	**Reaction Force Vector**	**Reaction Moment**	**Reaction Moment Vector**	**Associated Bodies**
"Fixed Support"	Fixed Edge	23,930.98 N	[-0.77 N x, 2,801.76 N y, 23,766.41 N z]	37.57 N·m	[0.0 N·m x, 37.31 N·m y, -4.37 N·m z]	*"PART2_1"*
"Fixed Support 2"	Fixed Edge	23,911.85 N	[0.77 N x, 2,633.35 N y, -23,766.41 N z]	1.84 N·m	[6.6×10^{-13} N·m x, -1.7 N·m y, 0.69 N·m z]	*"PART1_1"*

D.5.9 "Solution"

"Solution" contains the calculated response for *"Model"* given loading conditions defined in *"Environment"*.

It was selected that the program would choose the solver used in this solution.

D.5.10 Structural Results

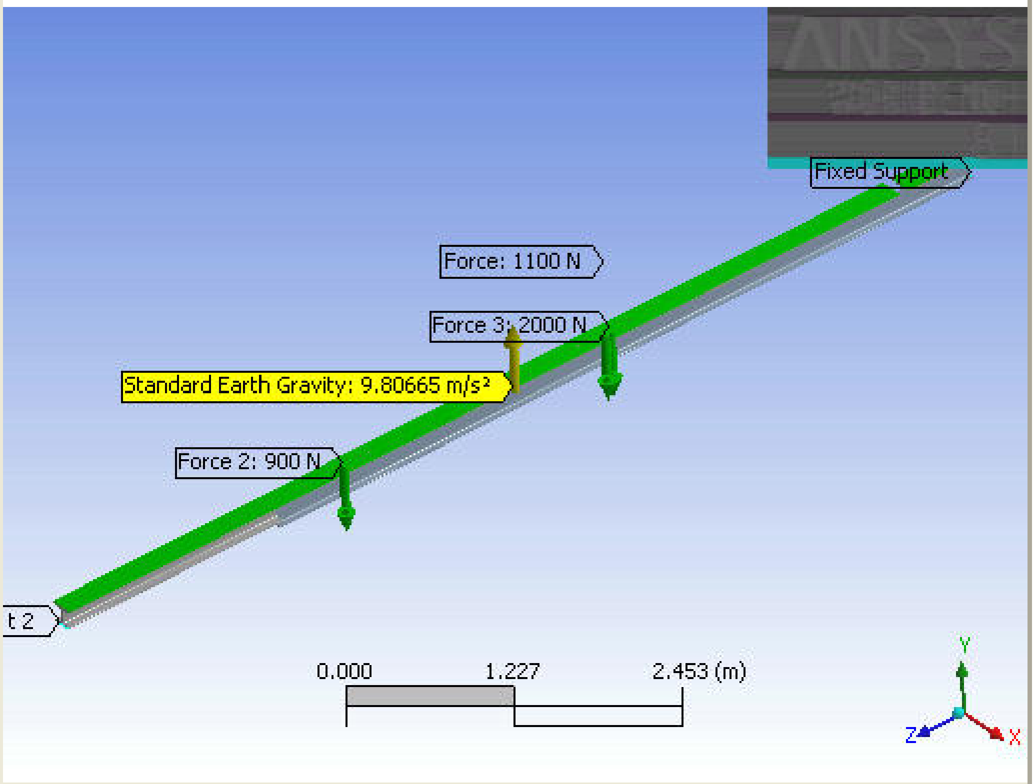
Values

Name	Figure	Scope	Minimum	Maximum	Alert Criteria
"Total Deformation"	A1.2	All Bodies In *"Model"*	0.0 m	8.2×10^{-3} m	None
"Equivalent Stress"	A1.3	All Bodies In *"Model"*	26,748.8 Pa	4.9×10^{7} Pa	None

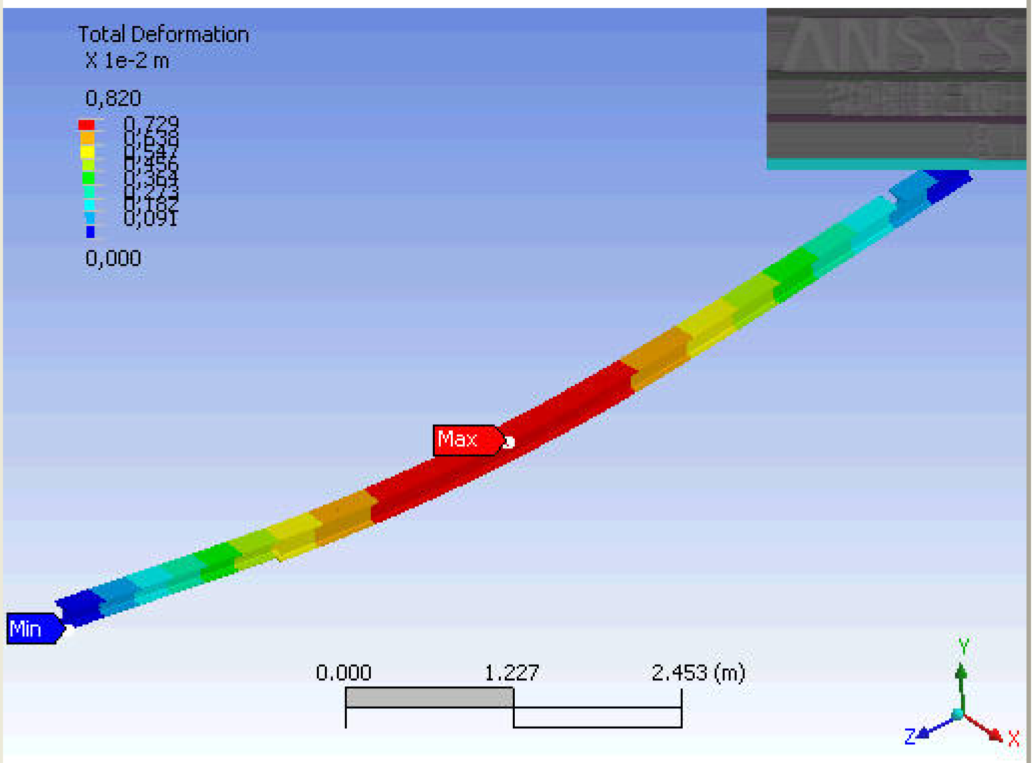
- Convergence tracking not enabled.

D.5.11 Scenario 1 Figures

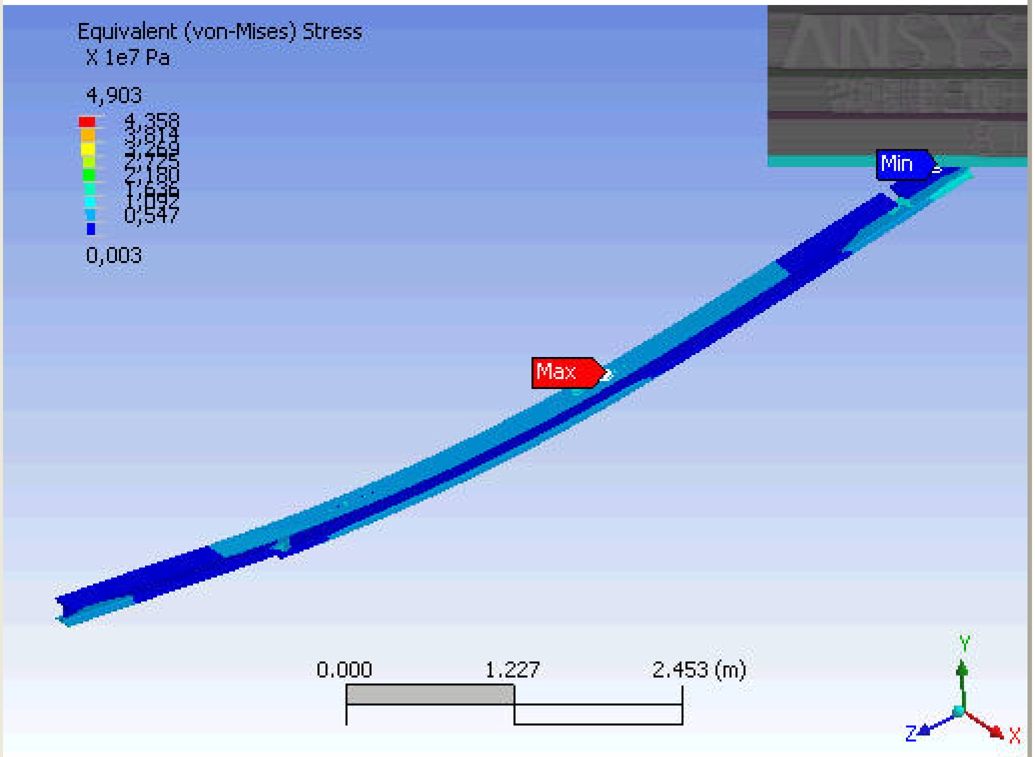
***"Environment"* Geometry**



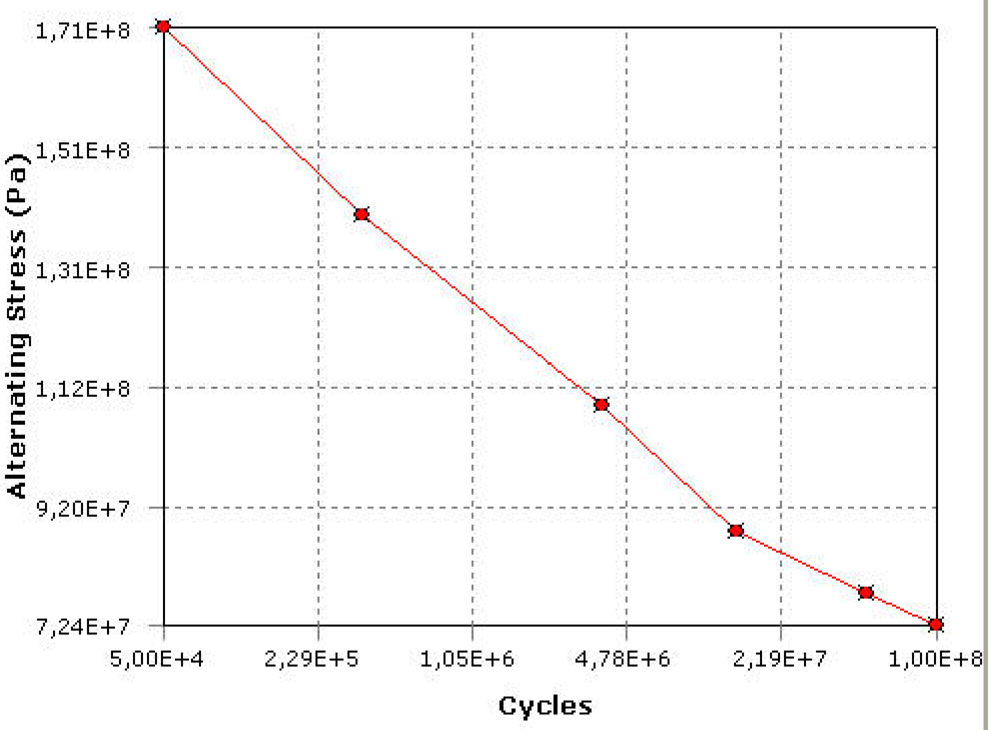
"*Total Deformation*" Contours



"Equivalent Stress" **Contours**



Alternating Stress vs. Cycles



E COST OF THE PROJECT

The cost of this project is expressed in terms of the cost of the work done by myself.

In this period of 16 months my responsibilities have been devised between the general planning activities and the progress of the TANs installation. In this project 900 hours have been worked that could be divided as follows:

700 h engineering	36, 00 €/h	25 200,00 €
100 h program + simulation	30, 00 €/h	3 000,00 €
100 h typing	15, 00 €/h	1 500,00 €
TOTAL		29 700,00 €

Table E.1. Cost of the project

The client of this project has been the CERN and particularly the LHC project. Apart from the costs shown above they should also be considered the amortization of the machinery needed for the tests and underground transport, and the labor needed to install the four targets TAN in place. Unfortunately these costs have to remain secret until the end of the LHC project.

The cost of the whole project has been considered by the client as acceptable since no unexpected troubles during the process of installation have been found for the moment. The cost-effectiveness of this project will be evaluated by the IC-PL team ones the 4 TANs are aligned and interconnected to the adjoining elements.