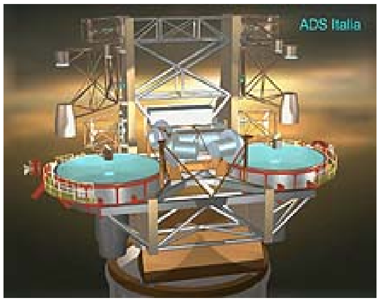
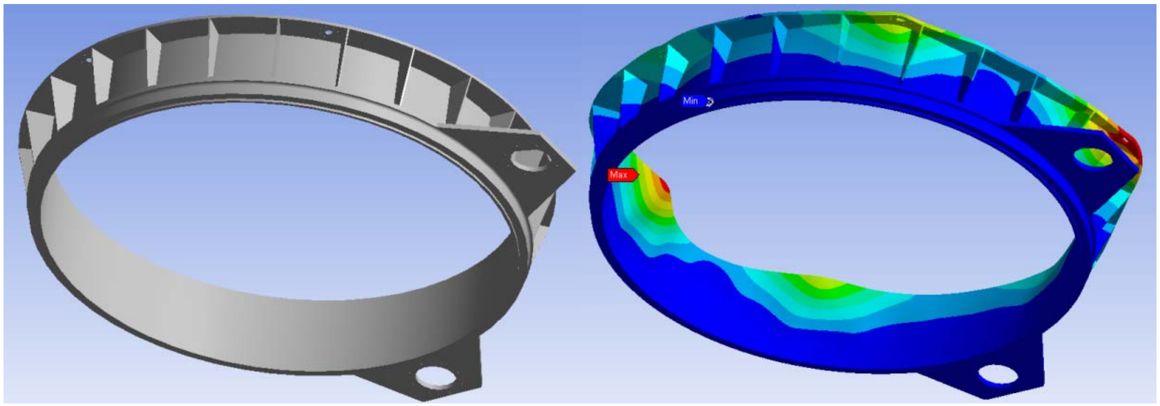


LBT PROJECT 2x8.4m TELESCOPE
Doc.No. : 674s008 Issue : a Date : 19 March 2007

LBT PROJECT

2 X 8.4m OPTICAL TELESCOPE



Instrument Rotator and Cable Chain

Instrument Support Structure Analysis

	Signature	Date
Prepared	Thomas Hair	03/19/2007
Reviewed	Robert Meeks	
Approved		

1 Revision History

Issue	Date	Changes	Responsible
a	05 March 07	First issue	T.Hair, R.Meeks

2 Table of Contents

1 REVISION HISTORY 2

2 TABLE OF CONTENTS 3

3 ABOUT THIS DOCUMENT 4

3.1 PURPOSE 4

4 MODEL DESCRIPTION 4

4.1 MESH 4
4.2 LOADS AND SUPPORT 6

5 RESULTS 6

5.1 ANALYSIS RESULTS SUMMARY 6
5.2 MODEL TOTAL DEFORMATION RESULTS 7

6 GENERATED ANSYS REPORT 8

6.1 SUMMARY 9
6.2 INTRODUCTION 9
6.3 SCENARIO 1 10
6.3.1 Gregorian Instrument Support Structure Analysis 10
6.3.2 Mesh 10
6.3.3 “Environment” 10
6.3.4 Structural Loading 10
6.3.5 Structural Supports 10
6.4 SOLUTION 11
6.5 APPENDICES 11
6.5.1 Definition of "Structural Steel" 11

3 About this document

3.1 Purpose

This document is to describe the analysis results from ANSYS about the deflection and stiffness of the Instrument Support Structure.

The results from the ANSYS analysis are to verify the stiffness of the Direct Gregorian rotator with the load applied when the telescope is horizon pointing.

4 Model Description

4.1 Mesh

Construction of an appropriate mesh accommodates applicable loads and supports giving results conducive of the actual model total deformation.

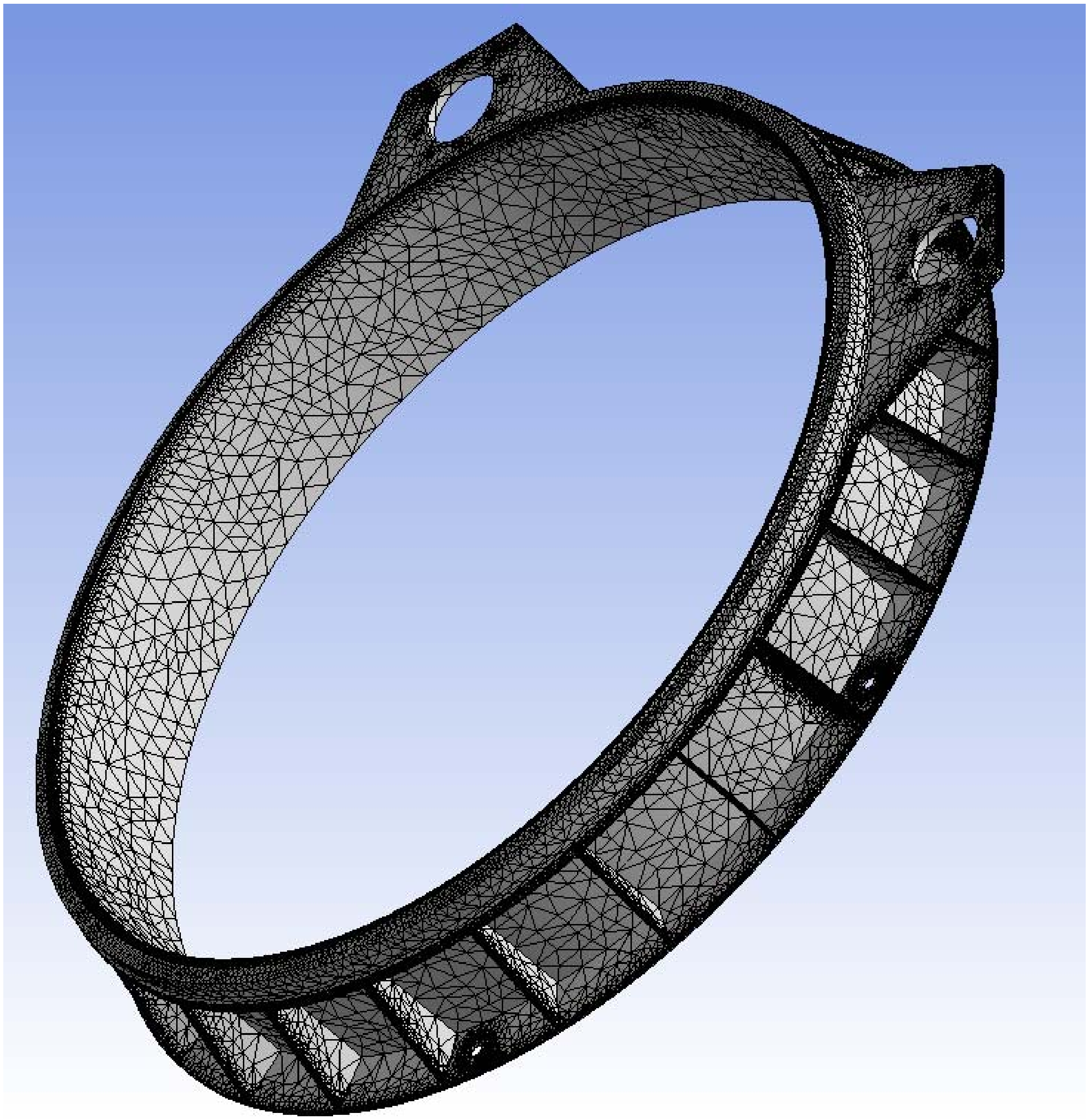


Figure 1 – Rotator Gallery Mesh

There are 180855 elements, and 372332 nodes in the mesh seen in Figure 1. The solid elements are primarily quadratic tetrahedrons and secondarily quadratic wedges. All parts in the assembly are “Structural Steel”. Definitions of “Structural Steel” are in section 6.5.1.

4.2 *Loads and Support*

A Force load was applied to the model according to a configuration that it would most likely to occur.

The load is 36000 N applied at 6 faces (highlighted in green in Figure 2) pointing in the direction of gravity as seen in Figure 2.

The fixed support is the circular face shown in green in Figure 2.

Any load could have been used instead of 36000 N. This is due to the linear relationship between load and deformation shown in Equation 1.

$$k = \frac{F}{\delta}$$

Equation 1 - Stiffness Calculation (stiffness is k, load is F, and deformation is δ)

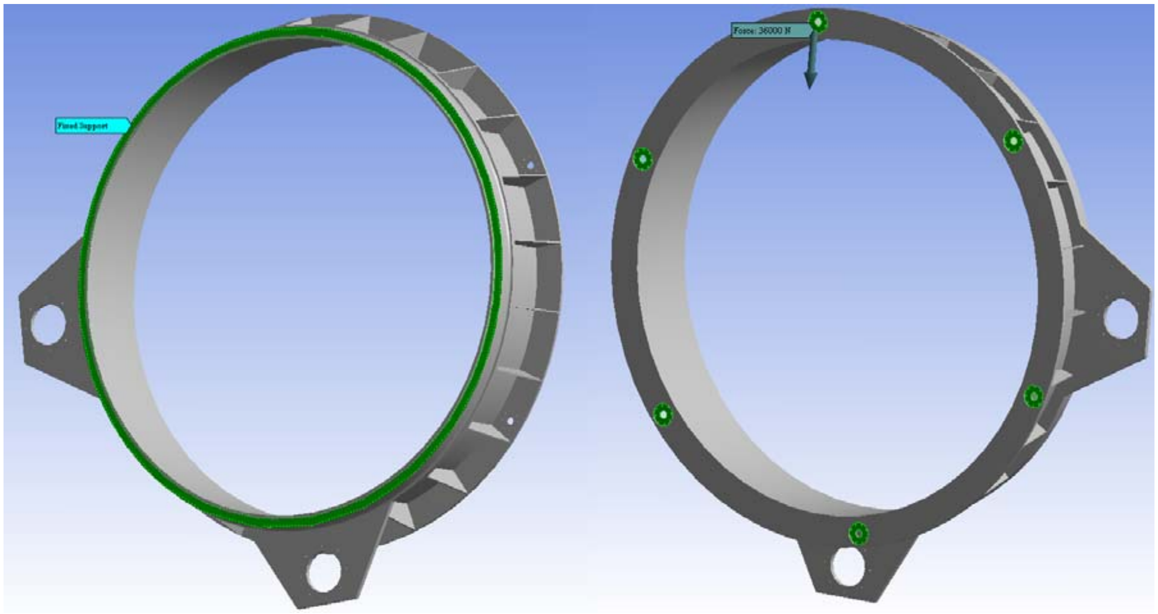


Figure 2 – Load and Fixed Support Selections

5 Results

5.1 *Analysis Results Summary*

The resultant maximum total deformation of the model is 23.2 microns at Horizon. This corresponds to a stiffness of 1555 N/micron.

The results obtained are from the calculations and analyses of the stiffness of the model.

The completed analysis returns information on the total deformation of the model. Stiffness is calculated by using Equation 1, the maximum deformation acquired from the completed analysis, and knowledge that the Load applied is 36000 N. This is shown as an example in Equation 2 using the results of the analysis of the model.

$$k = \frac{F}{\delta} = \frac{36000N}{.0232m} = 1555.1\frac{N}{\mu m}$$

Equation 2 - Stiffness of Instrument Support Structure

5.2 *Model Total Deformation Results*

Deformation of the model indicates a maximum deformation of 23.2 microns at Horizon as seen in Figures 3. This corresponds to a stiffness of 1555.1 N/micron.

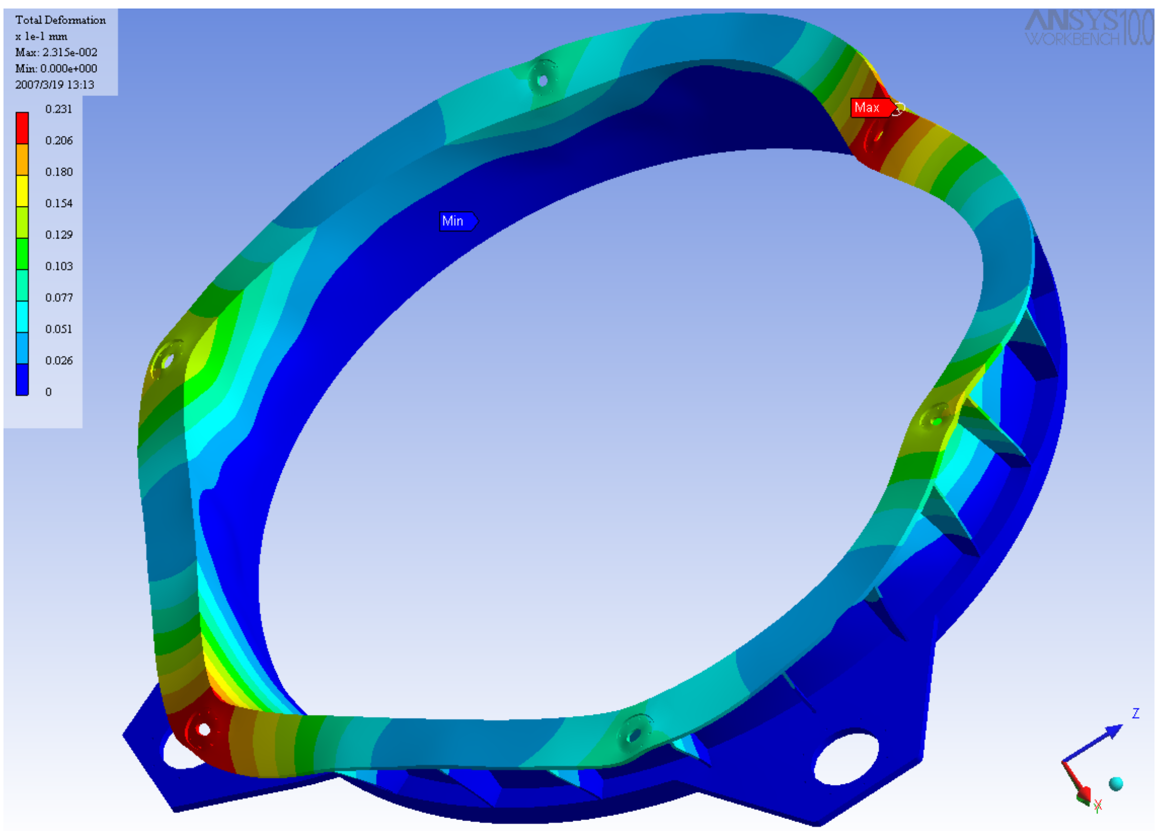


Figure 3 – Total Deformation of Instrument Support Structure

6 Generated ANSYS Report

Large Binocular Telescope Observatory

ANSYS Report

Gregorian Instrument Support Structure Analysis

Author

Thomas Hair

Subject

Gregorian Instrument Support Structure Analysis

Project Created

Friday, February 16, 2007 at 8:18:18 AM

Project Last Modified

Monday, March 19, 2007 at 12:15:10 PM

Report Created

Monday, March 19, 2007 at 12:18:26 PM

Software Used

ANSYS 10.0

6.1 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 Inventor part *"\\Lbtdw102\LBTO Home\thair\My Documents\Vault Working Folder\Projects\674 Gregorian Rotator\gregorian instrument support new.ipt"*.
- Considered the effect of 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.

6.2 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.

All values are presented in the *"Metric (mm, kg, N, °C, s, mV, mA)"* 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.

6.3 Scenario 1

6.3.1 Gregorian Instrument Support Structure Analysis

- The bounding box for the model measures 3,460.0 by 495.0 by 3,460.0 mm along the global x, y and z axes, respectively.
- The model has a total mass of 1,301.78 kg.
- The model has a total volume of 1.66×10^8 mm³.

6.3.2 Mesh

- *"Mesh"*, associated with *"Greorian Instrument Support Structure Analysis"* has a curvature/proximity value of 0.
- *"Mesh"*, has an element size of 100.0 mm.
- *"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 372332 nodes and 180855 elements.

6.3.3 "Environment"

Simulation Type is set to Static

Analysis Type is set to Static Structural

"Environment" contains all loading conditions defined for *"Greorian Instrument Support Structure Analysis"* in this scenario.

6.3.4 Structural Loading

Table 3.2.1.1. Structural Loads

Name	Type	Magnitude	Vector	Reaction Force	Reaction Force Vector	Reaction Moment	Reaction Moment Vector
"Force"	Surface Force	36,000.0 N	[0.0 N x, 0.0 N y,-36,000.0 N z]	N/A	N/A	N/A	N/A

6.3.5 Structural Supports

Table 3.2.2.1. Structural Supports

Name	Type	Reaction Force	Reaction Force Vector	Reaction Moment	Reaction Moment Vector
"Fixed Support"	Fixed Surface	36,000.0 N	[6.57×10^{-6} N x, -2.44×10^{-5} N y, 36,000.0 N z]	1.69×10^7 N·mm	[-1.69×10^7 N·mm x, 354,640.48 N·mm y, 6.37×10^{-3} N·mm z]

6.4 *Solution*

Solver Type is set to Program Controlled

Weak Springs is set to Program Controlled

Large Deflection is set to Off

- Convergence tracking not enabled.

6.5 *Appendices*

6.5.1 Definition of "Structural Steel"

Table A2.1. *"Structural Steel"* Constant Properties

Name	**Value**
Compressive Ultimate Strength	0.0 MPa
Compressive Yield Strength	250.0 MPa
Density	7.85×10^{-6} kg/mm³
Poisson's Ratio	0.3
Tensile Yield Strength	250.0 MPa
Tensile Ultimate Strength	460.0 MPa
Young's Modulus	200,000.0 MPa
Thermal Expansion	1.2×10^{-5} 1/°C
Specific Heat	434.0 J/kg·°C
Thermal Conductivity	0.06 W/mm·°C
Relative Permeability	10,000.0
Resistivity	1.7×10^{-4} Ohm·mm

Doc_info_start
Title: *Instrument Rotator and Cable Chain Instrument Support Structure Analysis*
Document Type: Report
Source: Steward Observatory
Issued by: Thomas Hair
Date_of_Issue: *19 Mar 2007*
Revised by:
Date_of_Revision:
Checked by:
Date_of_Check:
Accepted by:
Date_of_Acceptance:
Released by:
Date_of_Release:
File Type: MS Word
Local Name: *Instrument Support Structure Analysis*
Category: *600*
Sub-Category: *670*
Assembly: *674*
Sub-Assembly:
Part Name:
CAN Designation: *674s008*
Revision: *a*
Doc_info_end