LBT PROJECT

2 X 8.4m OPTICAL TELESCOPE

Instrument Rotator and Cable Chain

Instrument Support Structure Analysis

<table>
<thead>
<tr>
<th>Signature</th>
<th>Date</th>
</tr>
</thead>
<tbody>
<tr>
<td>Prepared</td>
<td>Thomas Hair</td>
</tr>
<tr>
<td>Reviewed</td>
<td>Robert Meeks</td>
</tr>
<tr>
<td>Approved</td>
<td></td>
</tr>
</tbody>
</table>
1 Revision History

<table>
<thead>
<tr>
<th>Issue</th>
<th>Date</th>
<th>Changes</th>
<th>Responsible</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>05 March 07</td>
<td>First issue</td>
<td>T.Hair, R.Meeks</td>
</tr>
</tbody>
</table>


## 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.
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 \( \delta \))*

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.

\[ k = \frac{F}{\delta} = \frac{36000 \text{N}}{0.0232 \text{m}} = 1555.1 \frac{N}{\mu m} \]

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

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 $1.66 \times 10^8$ mm³.

6.3.2 Mesh

- "Mesh", associated with "Gregorian 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 "Gregorian Instrument Support Structure Analysis" in this scenario.

6.3.4 Structural Loading

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Magnitude</th>
<th>Vector</th>
<th>Reaction Force</th>
<th>Reaction Force Vector</th>
<th>Reaction Moment</th>
<th>Reaction Moment Vector</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;Force&quot;</td>
<td>Surface</td>
<td>36,000.0 N</td>
<td>[0.0 N x, 0.0 N y, -36,000.0 N z]</td>
<td>N/A</td>
<td>N/A</td>
<td>N/A</td>
<td>N/A</td>
</tr>
</tbody>
</table>

6.3.5 Structural Supports

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Reaction Force</th>
<th>Reaction Force Vector</th>
<th>Reaction Moment</th>
<th>Reaction Moment Vector</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;Fixed Support&quot;</td>
<td>Fixed</td>
<td>36,000.0 N</td>
<td>[6.57×10⁻⁶ N x, 2.44×10⁻⁵ N y, 36,000.0 N z]</td>
<td>1.69×10⁷ N-mm</td>
<td>[-1.69×10⁷ N-mm x, 354,640.48 N-mm y, 6.37×10⁻³ N-mm z]</td>
</tr>
</tbody>
</table>
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>
<thead>
<tr>
<th>Name</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compressive Ultimate Strength</td>
<td>0.0 MPa</td>
</tr>
<tr>
<td>Compressive Yield Strength</td>
<td>250.0 MPa</td>
</tr>
<tr>
<td>Density</td>
<td>7.85×10⁻⁶ kg/mm³</td>
</tr>
<tr>
<td>Poisson's Ratio</td>
<td>0.3</td>
</tr>
<tr>
<td>Tensile Yield Strength</td>
<td>250.0 MPa</td>
</tr>
<tr>
<td>Tensile Ultimate Strength</td>
<td>460.0 MPa</td>
</tr>
<tr>
<td>Young's Modulus</td>
<td>200,000.0 MPa</td>
</tr>
<tr>
<td>Young's Modulus</td>
<td>200,000.0 MPa</td>
</tr>
<tr>
<td>Young's Modulus</td>
<td>200,000.0 MPa</td>
</tr>
<tr>
<td>Young's Modulus</td>
<td>200,000.0 MPa</td>
</tr>
<tr>
<td>Thermal Expansion</td>
<td>1.2×10⁻⁵ 1/°C</td>
</tr>
<tr>
<td>Specific Heat</td>
<td>434.0 J/kg-°C</td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>0.06 W/mm-°C</td>
</tr>
<tr>
<td>Relative Permeability</td>
<td>10,000.0</td>
</tr>
<tr>
<td>Resistivity</td>
<td>1.7×10⁻⁴ Ohm-mm</td>
</tr>
</tbody>
</table>