Integrated modeling for the GPi flexure sensitive structure

Darren Erickson*[a], Scott Roberts[a], John S. Pazder[a], J. Murray Fletcher[a]
[a]National Research Council - Canada, Herzberg Institute of Astrophysics
5071 W. Saanich Rd, Victoria, BC, Canada V9E 2E7

ABSTRACT

The Gemini Planet Imager (GPi) is comprised of three main opto-mechanical systems: the Adaptive Optics (AO) system, the Calibration (CAL) system, and the Integral Field Spectrograph (IFS). Each of these subsystems are built and aligned independently, and then integrated into the final instrument. A truss framework called the Flexure Sensitive Structure (FSS) has been designed to locate each optical subsystem within the instrument, utilizing kinematic bipods to eliminate distortion due to flexure and thermal changes.

Due to the distributed nature of the optical system, an end-to-end opto-mechanical modeling approach is taken using the NRC Integrated Model (NRCIM). This set of numerical tools was originally developed to support the Canadian VLOT and TMT telescope studies. The instrument structural response is calculated using a commercial finite element package; and the 6 degree-of-freedom rigid body motions of the optical elements are then passed to an optical model. Ray-tracing is performed to determine the line-of-sight errors at numerous critical focal planes and pupil planes. Disturbances to the system include gravity induced flexure and thermal distortions. Optical compensation using a combination of closed-loop feedback and open-loop models are then applied using steering mirrors to improve the line-of-sight figures of merit. Finally, these figures of merit are compared against the system optical error budget to assess the overall performance of the opto-mechanical system.

Keywords: Integrated modeling, NRCIM, Gemini Planet Imager, GPi, National Research Council, NRC

1. INTRODUCTION

This paper addresses two important aspects of the opto-mechanical performance of the GPi instrument. The first part presents the mechanical design and analysis of the Flexure Sensitive Structure (FSS), including its discrete opto-mechanical subsystems. The analysis uses finite element methods to study the flexure due to gravity and thermal distortion, as well as the vibration response of the system to a known input. The latter part of the paper introduces the NRC Integrated Model, and describes how it is used to assess the optical alignment at surfaces. The Integrated Model uses MATLAB to tie together the structural response and optical prescription, and generates true optical figures of merit at critical surfaces. A comprehensive overview of the GPi instrument can be found in reference [1].

2. OPTO-MECHANICAL DESIGN

2.1 Overview

The GPi instrument is comprised of two main sub-assemblies: the External Frame Structure (EFS), and the FSS. The EFS supports the electronics cabinets, houses the light-tight cover panels, and provides features for lifting and handling the instrument. The FSS resides inside the EFS, and includes the main optical subsystems and a stiff framework to support them (see Figure 1). The FSS and EFS were designed as two independent structures so that the additional mass of the EFS does not affect the flexure performance of the optical subsystems, and so that the vibration sources inside the electronics enclosures do not couple directly into the optical subsystems.

*darren.erickson@nrc-cnrc.gc.ca; phone 1 (403) 266-3477; http://hia-iha.nrc-cnrc.gc.ca/

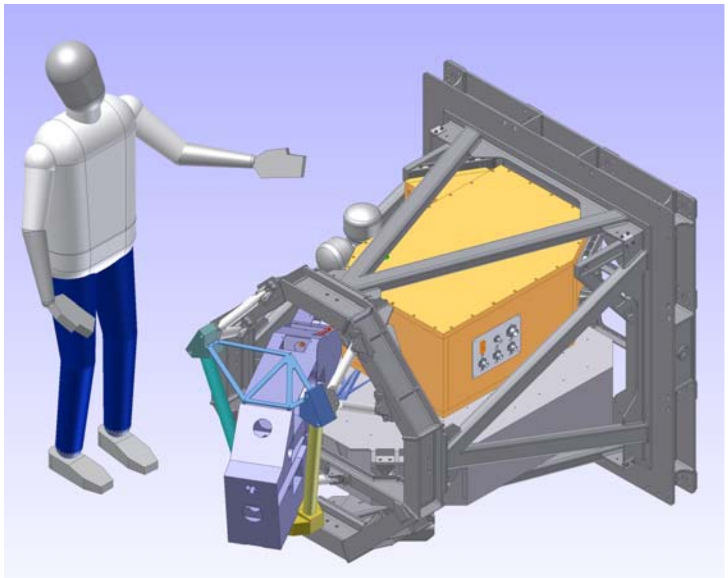

Figure 1: Flexure Sensitive Structure

2.2 Optical subsystems

Within the FSS, there are three opto-mechanical subsystems: the Adaptive Optics (AO) system, the CALibration system (CAL), and the Integral Field Spectrograph (IFS). A fourth optical system, the coronagraph, has three elements which are located throughout the instrument (one in each of the subsystems mentioned above). The AO system consists of a pair of off-axis parabola relays, forming pupils onto two deformable mirrors operating as a "woofer" and "tweeter". It also includes a high speed, spatially filtered Shack-Hartmann wavefront sensor taking visible-light measurements of the incoming wavefront. The calibration system uses an interferometer to measure slowly varying non-common path errors which are not sensed by the AO system. The IFS is the ultimate science instrument in the system. It is a lenslet based spectrograph, capable of producing data cubes (images and spectra) of extrasolar planets.

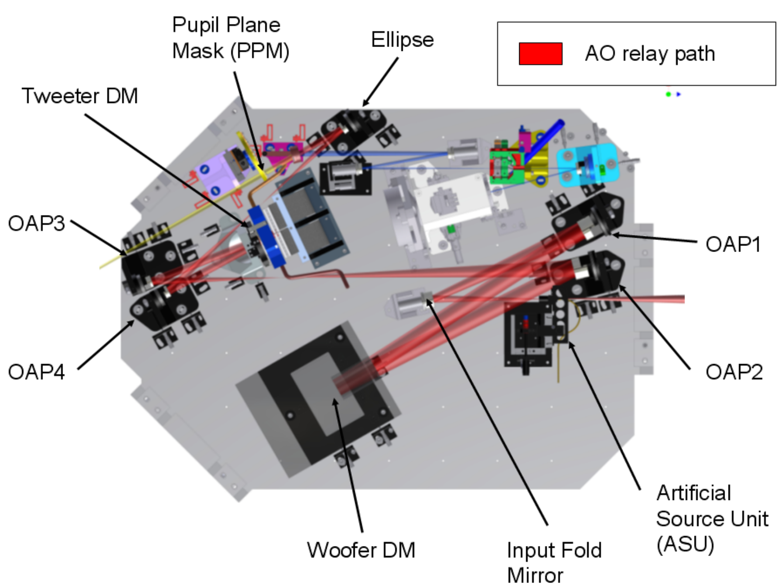

Figure 2: AO system overview

2.3 Flexure Sensitive Structure and bipod struts

The FSS structure must support the weight of the optical systems and maintain their relative alignment while subjected to changes in gravity vector orientation and changes in ambient temperature. To achieve this stability, the FSS is constructed as a structural steel weldment. The opto-mechanical subsystems are aluminum, and are attached to the FSS framework using kinematic bipods. Each subsystem has three pairs of bipods, providing exactly 6 constraints. By kinematically constraining each of the opto-mechanical subsystems, flexure of the FSS framework does not cause distortion in the optical systems (only rigid-body motion), and dissimilar materials can be used to optimize stiffness and weight in each component.

2.4 Alignment specifications

The alignment specifications of the optical systems are very tight to achieve the required image quality and contrast ratios for planet detection. The difficulty of these specifications is compounded by the fact that each opto-mechanical subsystem is constrained as an independent unit, with its own unique response to gravity, thermal changes and vibration. As an example, the Pupil Plane Mask (PPM) within the coronagraph has an aperture stop which is undersized by 1% of its diameter (12 mm). In order for light not to leak through the coronagraph, the beam which illuminates this aperture must not wander by more than 60 μm across the face of the mask. This top-level specification is partitioned into components for gravity flexure, thermal flexure, measurement and control errors, and vibration.

3. MECHANICAL ANALYSIS

3.1 Finite Element Model

A detailed finite element model of the FSS has been constructed using the ANSYS Parametric Design Language. This model includes the structural components, AO optical system, CAL optical system and IFS (Figure 3). This model has been carefully verified using a number of techniques, and is believed to be an accurate model of the FSS structural mechanics.

The current modeling of optical surfaces in GPi is built upon one basic assumption: that the optic surface and mount are stiff compared to the response of the optical bench to which they are attached. Given the high stiffness and low mass of the actual mounts, it is believed that this assumption is valid and appropriate for simplifying the modeling task (see Figure 4). In the future, as the designs of the optical mounts are incorporated into the finite element model, the analysis will be rerun to ensure that the design of the mounts has not significantly changed the system performance.

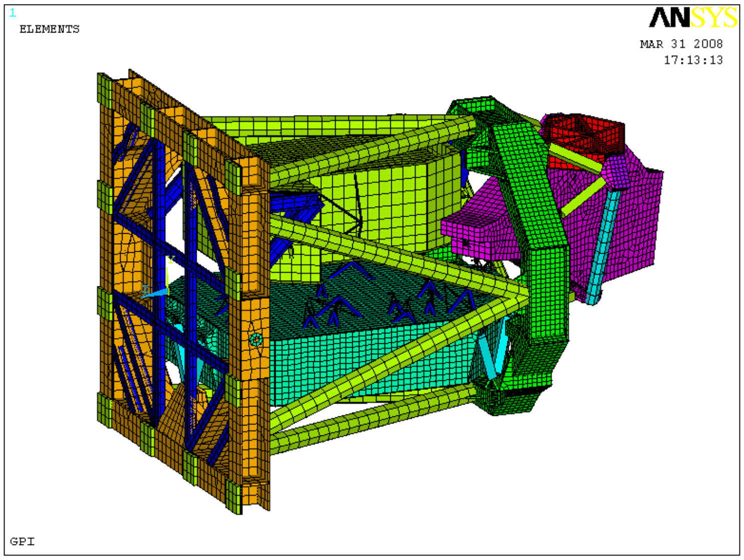

Figure 3: FSS finite element model

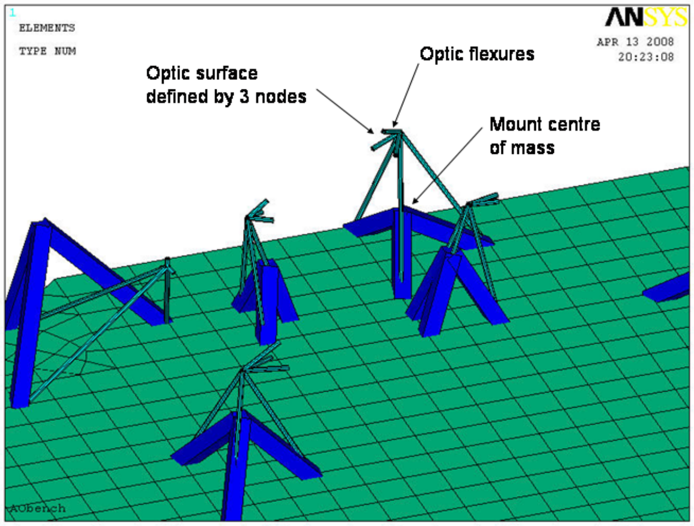

Figure 4: Optical Element Representation in FEM

3.2 Gravity Flexure

A set of 119 independent gravity vector orientations was developed to simulate performance over all possible combinations of the Gemini telescope mounting ports, elevation angles and Cassegrain rotator angles. These local gravity vectors were then used to determine the flexure response of the instrument. Bulk flexure magnitudes were small, showing maximum deflections in the X-Y-Z principal directions of 92 μm, 70 μm, and 40 μm respectively. In addition to bulk motions, the 6-dof deflections of each optical surface were also calculated. These motions were used to calculate optic surface flexure rates, assuming the telescope moves at the maximum rate of 15 degrees per hour. Typical rates for components on the AO system were less than 10 μm/hour.

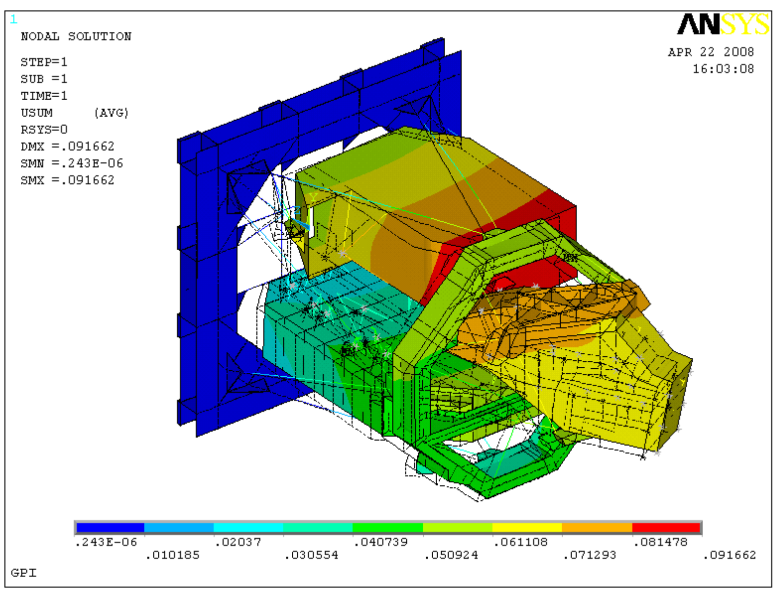

Figure 5: Gravity flexure (X direction)

3.3 Thermal Flexure

The finite element model was also used to study the effects of ambient temperature variations between +20°C and -5°C. As expected, the instrument demonstrated a linear response, contracting towards the central optical axis and towards the mounting interface on the telescope. The magnitude of the response was approximately 0.7 mm (max) measured at the back edge of the CAL system.

3.4 Vibration Response

Vibration is a concern for the performance of GPi due to the low bandwidth of the CAL system measurements of non-common path errors. The most critical optic in the system in terms of vibration response is the CAL Focal Plane Mask. The woofer tip/tilt platform is controlled to deliver the image at the centre of this mask, but is updated by pointing offsets from the CAL at a rate of only 1 Hz. Although this is fast enough to compensate for flexure errors due to gravity and thermal distortion, it is not fast enough to correct for vibration. The image alignment specification is 6 μm RMS.

Modal analysis was performed as the first step of the vibration analysis. The goal of the FSS structural design was to have the first natural frequency above 100 Hz. Due to the high mass of the FSS and opto-mechanical subsystems (970 kg), 5 modes below 100 Hz were found:

Mode	Freq (Hz)	Description
1	60	X-direction swaying of main truss members
2	71	Y-direction swaying of main truss members
3	82	Rigid-body translation of IFS (Z dir)
4	85	Torsion of AO bench (about Z axis)
5	99	Rigid-body translation of AO bench (Z dir)

Based on previous experience with Gemini instruments, it is assumed that the primary source of vibration for the instrument will come from the telescope itself, through the mounting face. Accelerometers were used by Gemini staff to create a time-history of acceleration at the mounting face which was processed and turned into an acceleration power spectral density (PSD). The square-root of the area under this PSD has also been calculated, showing an RMS acceleration of 0.00256 g (see Figure 6). Given the structural natural frequencies in the 60-80 Hz range, the large amount of input power in the same band is a serious concern due to resonances.

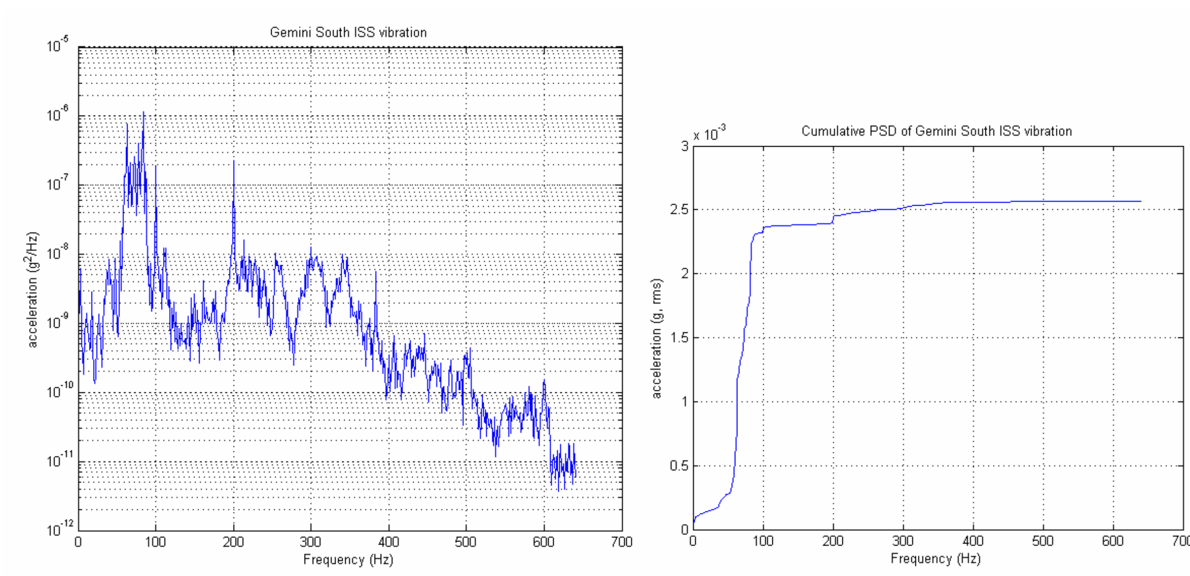

Figure 6: Vibration PSD and cumulative RMS

This PSD was applied to the base of the instrument, equally in the X, Y and Z directions. The ANSYS spectrum analysis capability was then used to calculate acceleration and displacement response PSDs at each of the optical surfaces, for each degree of freedom (eg, X-Y-Z directions). This analysis assumed a constant damping ratio of ζ=0.5%. Not surprisingly, the displacement response PSDs show a large gain in motion at the PPM and FPM compared to the base of the instrument within the 60-100 Hz frequency range (see Figure 7). Despite these unwanted resonances, the RMS motions were found to be small (2.5 μm RMS at the FPM – Y dir), which is within the specification of 6 μm RMS.

Disp. (mm, RMS)	PPM	FPM
X dir	0.0012	0.0017
Y dir	0.0013	0.0025
Z dir	0.0009	0.0002

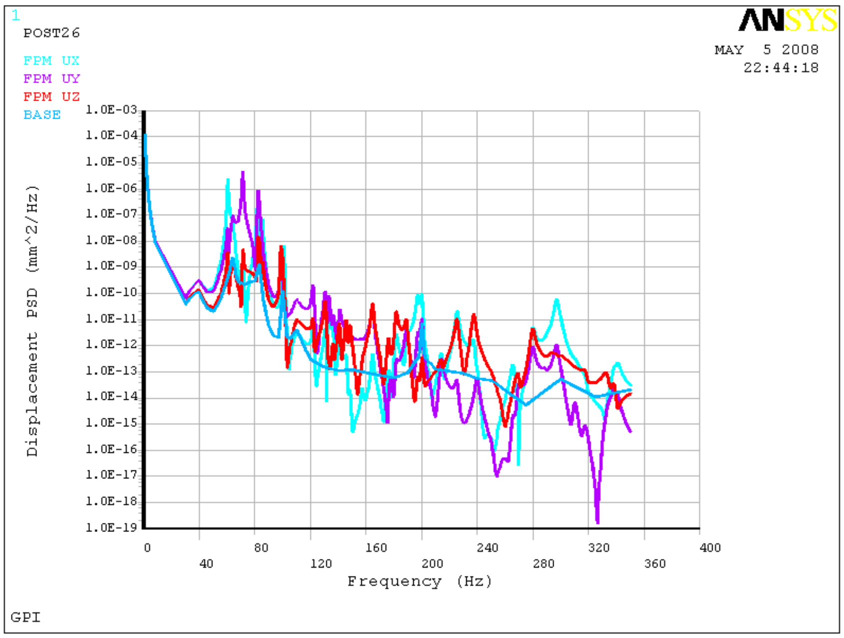

Figure 7: Displacement response PSD at FPM

4. NRC INTEGRATED MODEL

4.1 Structure of the IM

The NRC Integrated Model (NRCIM) toolset can be used to analyze the effects of gravity and thermal induced deflections on a wide variety of opto-mechanical systems. A typical objective of the opto-mechanical system design process is to minimize image quality degradation due to uncorrectable deflections of the optics as a result of internal or external disturbances, while ensuring that the overall system meets several requirements including image quality, line of sight position and jitter, and constraints including available control system actuator dynamic range. These design and performance criteria and constraints are not readily calculated within a finite element program, and are tedious and prone to error when manually calculated. The NRCIM toolset, implemented in MATLAB with interfaces to ANSYS and ZEMAX, automates these analyses, allowing a much higher degree of analysis and optimization than previously possible. In an iterative design process, the results can be fed back to ANSYS finite element software for evaluation and design optimization. The NRCIM toolset has been applied to many problems, including the optimization of the VLOT telescope design [2][3], the analysis of the TMT telescope structural model [4], and recently to characterize the GPi performance.

In the GPi analysis, NRCIM was used to translate ANSYS Finite Element Model (FEM) displacements into optical surface motions in ZEMAX. MATLAB was used to simulate the control of the pointing and centering mirrors that compensate the opto-mechanical misalignments within the system. Pupil and image motions after compensation were reported for more than 120 different load cases for the GPi CDR study. In the future we also plan to study the dynamic response of the GPi system.

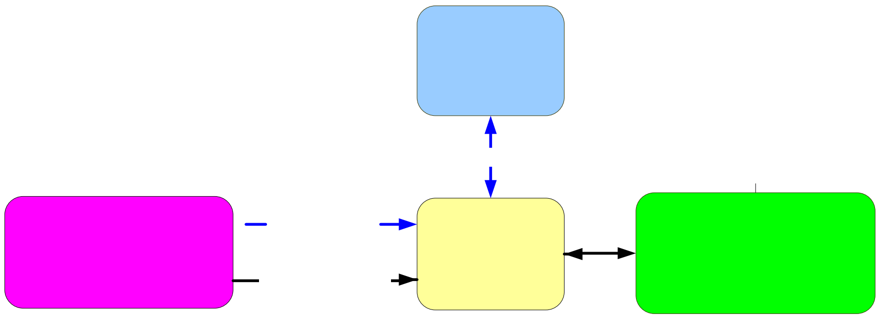

Figure 8: NRCIM Model Structure

The NRCIM model structure is depicted in Figure 8. The NRCIM toolset includes many MATLAB routines that enable interfacing with ANSYS and ZEMAX, and manipulation of the ZEMAX model through a DDE interface. Data structures including coordinate system transforms and details of the interfaces to each of the FEM and optical models are generated in a one-time initialization process, and used during the analysis process.

4.2 Data Structures

The NRCIM data structure contains information identifying the optical surfaces that will be manipulated and about how to link together the ANSYS FEM and ZEMAX optical models. This data structure is unique for each system being modeled, and it controls how all the other NRCIM data structures are generated. It includes identifiers for ANSYS nodes and ZEMAX surfaces, as well as information about how to generate and calculate the coordinate system transforms between the ANSYS and ZEMAX models. In the NRCIM structure the references to the FEM and optical models are by an interface using component names. This enables a simple process of updating the models and regenerating the data structures.

At the heart of the NRCIM toolset are a set of data structures *(IM*, *MA*, *MZ*, and *CST*) that define the characteristics of the system being modeled. The *IM* data structure includes high level meta-data regarding the analysis being performed including names of configuration files, optical prescription, and units. The *IM* data structure contents are defined from an input file called as an argument during the data structure initialization process.

The *MA* data structure contains information about the MATLAB to ANSYS interface, including node numbers that are used to establish coordinate systems and transforms. This data structure is defined based on the information in the NRCIM data structure, and through interrogation of the ANSYS model database via automatically generated APDL script files that are run in ANSYS and then read back into MATLAB.

The *MZ* data structure contains information about the MATLAB to ZEMAX interface, including the row numbers and types of surfaces in the optical model. This information is based on information in the NRCIM data structure, and through interrogation of the ZEMAX model via the DDE interface from MATLAB.

The *CST* data structure contains the coordinate transforms for the entire system, including the coordinate transform between the global ZEMAX and ANSYS coordinate systems, and the transforms from global to local surface coordinate systems in ZEMAX and from global to local coordinate systems in ANSYS.

4.3 Coordinate systems

The NRCIM model constructs a set of coordinate systems in data structures within MATLAB. The following coordinate systems are utilized by NRCIM.

- **G** is the base coordinate system in ANSYS. All nodal coordinates are defined in this coordinate system.
- **Z** is the base coordinate system in ZEMAX.
- $\mathbf{C}_n$ is the coordinate system used to define the structural motion in ANSYS that defines the motion of the n^{th} optical element in ZEMAX. Perturbations to $\mathbf{C}_n$ are calculated based on nodal displacements in the ANSYS model.
- $\mathbf{M}_n$ is the coordinate system used to define the position of the n^{th} optical element in ZEMAX. Commands to move a surface in ZEMAX are Euler angles in this coordinate system.

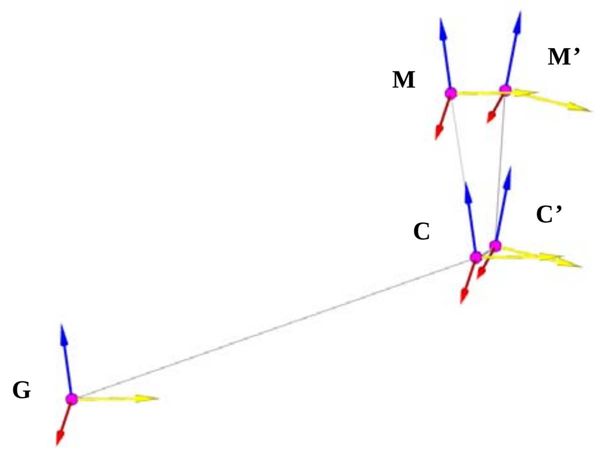

Figure 9: Relationship between NRCIM coordinate systems

NRCIM uses the coordinate transform between the **C** and **M** at each surface to determine the appropriate optical motions to perform in the ZEMAX model. The **C** and **M** frames are assumed to be rigidly connected, so it is important that the perturbation of **C**, designated **C**', resulting in a motion of the optical surface, **M**', be representative of the motion of the optical element. This is illustrated in Figure 9. Note that **C** and **M** do not have to be co-located and can have any orientation relative to each other. The NRCIM code takes all these transforms into account in the analysis.

In the finite element model, the motion of the surface is represented as a coordinate transform between the nominal and perturbed location of a set of defining nodes. Three nodes forming an equilateral triangle (called a 'triad') are used to define each FEM coordinate system. The position of the 3 nodes is used to define the **C** coordinate system for that surface, and the motion of the nodes is used to determine the perturbation of the optical surface under the applied load condition. The 3 nodes must have their nodal coordinate systems rotated into the coordinate system that is defined by the triad.

4.4 NRCIM Run Initialization Process

The following one-time steps are necessary to prepare for an NRCIM analysis:

1. Prepare the ZEMAX model for the interface to NRCIM by adding coordinate break surfaces with identifiers to the prescription. These identifiers are later read from the prescription via the MATLAB DDE interface to determine the locations within the prescription where optical perturbations are made and results are analyzed.

2. Prepare the ANSYS model by adding component names to identify the interface nodes (triads) that define the ANSYS coordinate systems.

3. Write the IM.ZGName file, which defines the coordinate transform between the ANSYS and ZEMAX global coordinate systems. This is needed to reference the coordinates in the ANSYS model to those in the ZEMAX model.

4. Write the NRCIM data structure definition file, which relates the ANSYS and ZEMAX surfaces, and ultimately the coordinate systems, to each other. It also contains information about ZEMAX surface types and methods for determining the CSTs from the ANSYS model.
5. Write the *IM* data structure input file, which defines the input and output file names, and the ZEMAX prescription name for the analyses.
6. Create the FEA data extraction script, which is generated from the NRCIM data structure. This is run in ANSYS to extract the interface node numbers and locations form the model.
7. Run the NRCIM initialization routine, which reads the ANSYS data interface file, creates a link to ZEMAX and then generates and saves the *IM*, *CST*, *MA*, *MZ*, and NRCIM data structures.
8. Create MATLAB functions to implement control loops within the optical model. The NRCIM Data Structures provides hooks that can be used to interface with and read back ray trace information from ZEMAX and manipulate the system from MATLAB. Complex control loops can be created in the MATLAB environment.
9. Once the initialization is complete, ANSYS is run to extract nodal displacements for each load case, and then the NRCIM is run to perturb and simulate the control of the ZEMAX model to produce the simulation results. This step can be repeated again and again for different analysis cases without repeating the above steps.

In the case of GPi, there are two sets of data structures with corresponding ZEMAX models, one for the science path, and one for the AO wavefront sensor path. The control loop tilts the input fold mirror and woofer DM to center the on-axis chief ray on tweeter DM and FPM within a tolerance of 0.1 μm using optimization of the ray trace results from ZEMAX in MATLAB. The NRCIM model loads and unloads each model as necessary to perform the analysis and close the control loop.

4.5 Validations

Simple Beam Model

In order to validate the basic NRCIM tools, a test case of a simple beam with two mirrors attached was constructed. ANSYS FEA and ZEMAX models were constructed for this case, and the NRCIM data structures were generated, and a simple gravity load case was analyzed. The structural model was independently calculated as a beam deflection, and the expected motions of the optics were calculated.

The ANSYS results were verified against the analytic solution to the model, and the calculated coordinate systems and their gravity induced perturbations were compared with the results calculated in NRCIM. The results were found to be in agreement. The major purpose of this test was to ensure that there were no errors in the calculation of the coordinate systems. This is a basic calculation, but coordinate system calculations are prone to error if not done carefully and verified.

Cross model validation

The Integrated Model is quite complex and thus it was felt that a second independent examination of the effects of gravity perturbations should be undertaken. The verification model was done entirely within the ZEMAX environment. Global perturbations from an FEA model were supplied in a table. The motions were displacements in X, Y and Z and tilts about the X, Y and Z axes.

An optical prescription of the AO path was modified to allow simple insertion of the global perturbations directly into the prescription. Since the original prescription was sequential and all of the optical surfaces were handled in local coordinates, further coordinate breaks were added to, in effect, switch to global coordinates.

The NRCIM analysis and hand-coded analysis were validated by comparing cases with FEA perturbations loaded, and the same cases with the optical feedback loop closed. Global positions of the optical elements and ray traces were used to debug both models until there was excellent agreement between the two. Differences in the on-axis chief ray intersection and global positions were at the micron level, which is small compared to the ray deviations at the surfaces. These insignificant residuals were attributed to differences in the FEA perturbation calculation and differences in the point of application of these perturbations in the models.

5. INTEGRATED MODELING RESULTS

5.1 Overview of gravity and thermal flexure

The NRCIM has been used to calculate the line of sight errors at the primary pupil and image planes within the GPi science path. The total (vector sum) amount of X-Y misalignment of the on-axis chief-ray on the surface of each optic was calculated for each of the 119 gravity flexure cases and the thermal cases. As expected, the response is smooth as the telescope moves in zenith angle, but shows large discontinuities when the zenith angle is set from 50° back to 0°, and the Cassegrain rotator is indexed by 60°. With only a few exceptions, the optical misalignments at the surfaces were less than 25 microns. The PPM and WFS lenslet array showed significantly less misalignment, with maximum values around 5 microns.

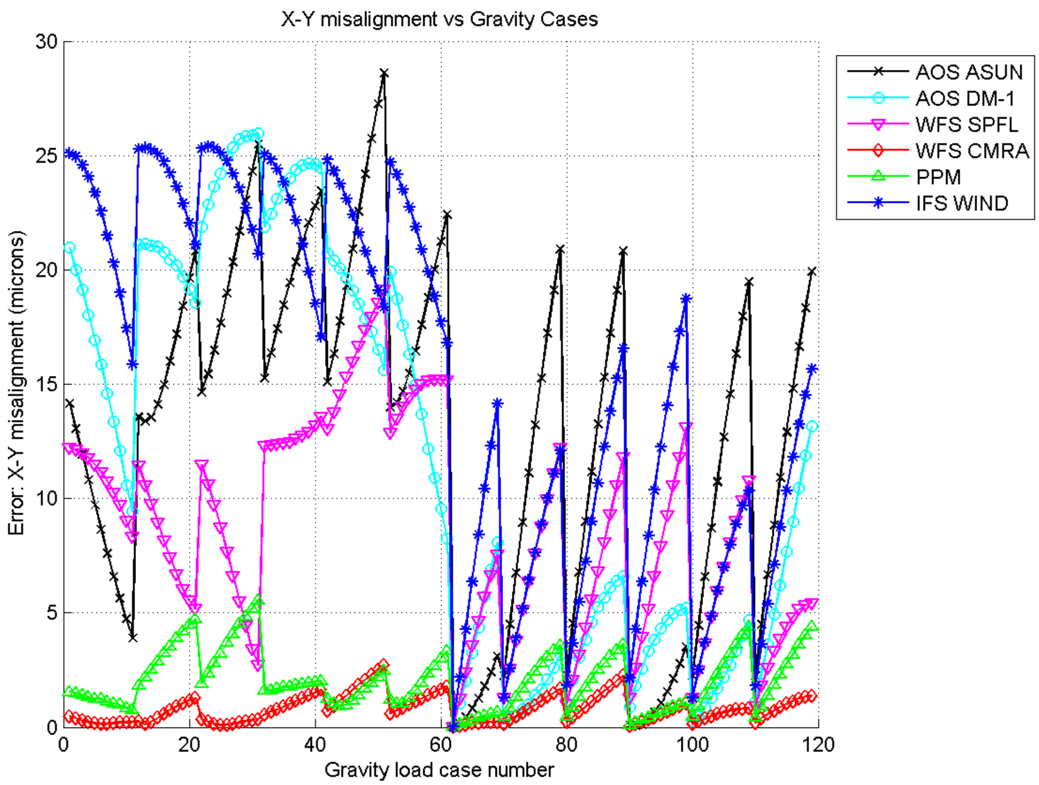

Figure 10: Optical misalignment (X-Y translation of chief-ray) due to gravity flexure

5.2 Pupil Centering

At the Pupil Plane Mask, the primary quantities of interest are the total (maximum) optical misalignment and the rate of change of this misalignment. The top-level specification for PPM alignment allows for a 60 μm deviation of the chief-ray. Within this top-level spec, 15 μm was allocated for errors due to flexure. Figure 11 shows that the maximum error due to flexure was below 6 μm and well within specification. The flexure data was used to calculate the rate of change of optical misalignment, assuming the maximum telescope tracking speed of 15°/hour. At the PPM, this rate was below 1.5 μm/hour (Figure 12).

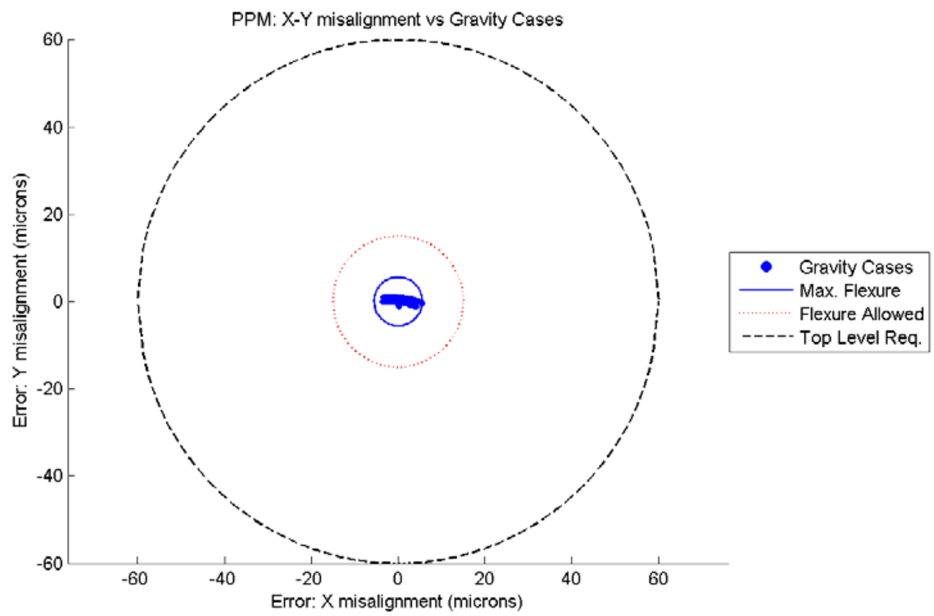

Figure 11: Pupil alignment at the Pupil Plane Mask

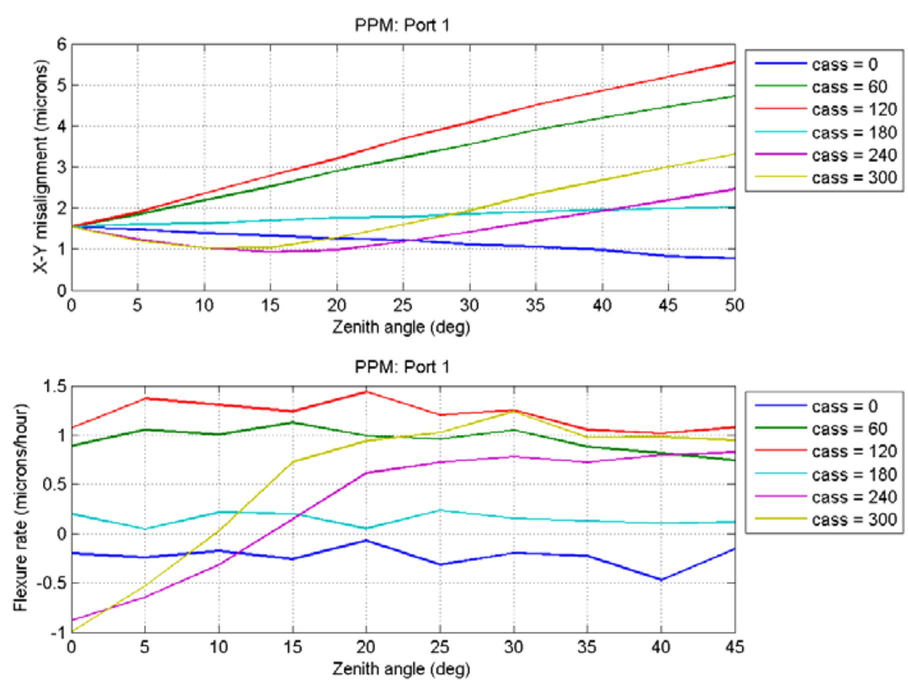

Figure 12: Rate of change of PPM alignment

6. CONCLUSIONS

The GPi Flexure Sensitive Structure is made up of discrete opto-mechanical subsystems, held together by a stiff truss framework. To achieve high image quality and contrast ratio, these subsystems must maintain very tight optical alignment specifications. To limit distortion of the optical systems, each is constrained using kinematic bipods.

Detailed mechanical analysis has been performed using finite element analysis. The total flexure at each optical surface was calculated for a wide variety of gravity and thermal load cases. In addition, the effects of vibration were studied through modal analysis and spectrum analysis using a realistic input acceleration PSD. The vibration response of a particularly critical optic (the CAL Focal Plane Mask) was found to be less than the spec of 6 μm RMS.

However, to fully assess the optical performance of the instrument, mechanical analysis is not sufficient. The NRC Integrated Model uses a suite of custom MATLAB routines to tie together the structural response from ANSYS with the optical prescription in ZEMAX. Motions of optical surfaces were used to perturb surfaces in ZEMAX, and closed-loop control of pointing and centering mirrors was implemented in MATLAB. Results show that optical misalignments at critical image and pupil planes are generally smaller than the purely mechanical motion and within specifications.

7. FUTURE WORK

We wish to extend the use of the NRCIM to include opto-mechanical analysis of structural dynamics and instrument vibration. One avenue of exploration is to generate time-domain simulations of the vibration response of the system, and analyze the nodal displacements at each time step as if they were static load cases (analogous to the gravity or thermal results presented above). Although computationally expensive, this technique would demonstrate how the optical alignment varies with time in a dynamically responding instrument.

REFERENCES

[1] Macintosh, B., et at, "The Gemini Planet Imager: from science to design to construction", Proc. SPIE Vol. 7015, paper 43, (2008).

[2] Roberts, S., Pazder, J., Fitzsimmons, J., Herriot, G., Loewen, N., Smith, M., Dunn, J., Saddlemyer, L. "Integrated Modeling of the Canadian Very Large Optical Telescope", Proc. SPIE Vol. 5382, p. 346-355, (2004).

[3] Pazder, J., Roberts, S., Smith, M., Lavigne, J.-F., McKenzie, H. "The optical modeling tools for the Canadian Very Large Optical Telescope Integrated Model", Proc. SPIE Vol. 5382, p. 404-412, (2003).

[4] Roberts, S., Sun, S., Kerley, D. "Optical performance analysis and optimization of large telescope structural designs", Proc. SPIE Vol. 5867, p. 200-211, (2005).