

# Structural Analysis, Modeling, and Testing of Multi-nut Joint for TriTruss Structure

Kyongchan Song<sup>1</sup>, Olive Stohlman<sup>2</sup>, Brace White<sup>1</sup>, and  
Ray Shimry Garatsa<sup>2</sup>

*NASA Langley Research Center, Hampton, VA 23681*

The Multi-nut joint, developed at NASA Langley Research Center (LaRC), was designed to assemble TriTruss modules for a doubly curved 65.6-foot telescope support structure. Axial and bending tests for a multi-nut joint in the TriTruss structure were conducted to characterize the structural performance of the multi-nut joint and to obtain the installation torque value of the attachment fastener. This paper describes the summary of the axial and bending tests of the multi-nut joint, along with the test results. The equivalent axial stiffness (EA) and equivalent bending stiffness (EI) of the multi-nut joint were characterized based on these test results and are presented in this report. Additionally, finite element models (FEMs) of the axial and bending test specimens were developed, and the predicted axial and bending responses of the multi-nut joint were compared with the test results. Test-analysis comparison results show that the predicted axial load-displacement response of the test specimen was within 5 percent of the test results, while the predicted load-bending response of the test specimen exhibited a consistent trend with the test data.

## I. Nomenclature

|             |   |                                          |
|-------------|---|------------------------------------------|
| <i>EA</i>   | = | Joint equivalent axial stiffness         |
| <i>EI</i>   | = | Joint equivalent bending stiffness       |
| <i>FEM</i>  | = | Finite element model                     |
| <i>iSAT</i> | = | in-Space Assembled Telescope             |
| <i>kip</i>  | = | kilopound per square inch                |
| <i>LaRC</i> | = | NASA Langley Research Center             |
| <i>LVDT</i> | = | Linear Variable Differential Transformer |
| <i>MTT</i>  | = | Material Testing Technology              |
| <i>PASS</i> | = | Precision Assembled Space Structure      |

## II. Introduction

The Precision Assembled Space Structure (PASS) project has the goal of developing a structural design and demonstrating an autonomous assembly method for a 65.6-foot telescope support structure. The structural shape is based on the in-space assembled telescope (iSAT) study [1], which details a 65.6-foot diameter telescope parabolic mirror built up from individual truss units called TriTrusses as shown in Fig. 1.

---

<sup>1</sup> Research Aerospace Engineer, Structural Mechanics and Concepts Branch, AIAA Member.

<sup>2</sup> Research Aerospace Engineer, Structural Dynamic Branch, AIAA Member.

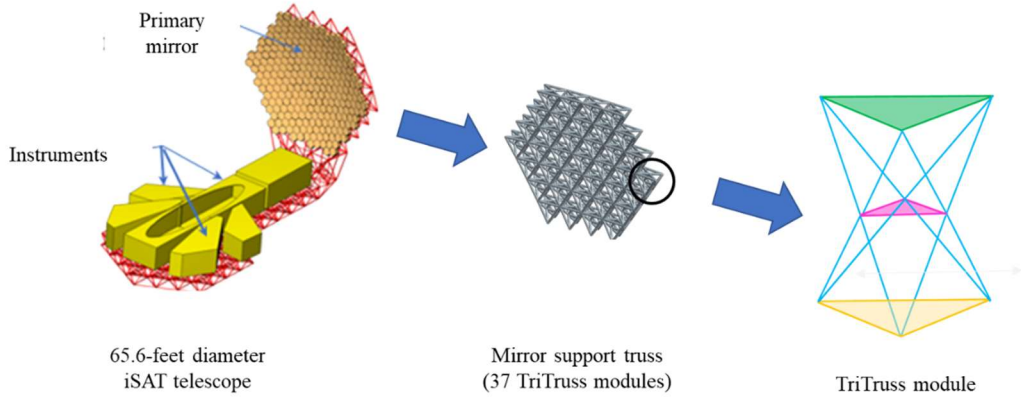


**Fig. 1 iSAT support structure and TriTruss module.**

The TriTruss [2-4] is an innovative structural module that has been developed by researchers at NASA Langley Research Center (LaRC) for near-zero-g (i.e., the gravitational force experienced in near Earth orbit) in-space-assembled telescopes and platforms for science and communications. Some of the innovative features of the TriTruss include compact packaging for launch, the option for staged packaging, and simple robotic deployment. Additionally, its capability for module-to-module robotic assembly and design versatility allows the TriTruss to be customized for specific applications [2].

For the PASS project, a doubly curved 65.6-foot diameter telescope support structure will be constructed by 37 identical wedge-shaped TriTruss modules. In order to achieve a doubly curved structure composed of uniform TriTruss modules, gaps are intentionally left at the nodes between connecting modules. These gaps are filled using uniquely shaped nodal connectors called multi-nuts, which connect adjacent modules using fasteners [5]. The benefits of having uniform TriTruss modules outweigh the structural penalties as it would greatly simplify assembly and lower manufacturing costs. A wedge-shaped TriTruss module and the multi-nut joint which connects corner nodes of three TriTruss modules are shown in Fig. 2.

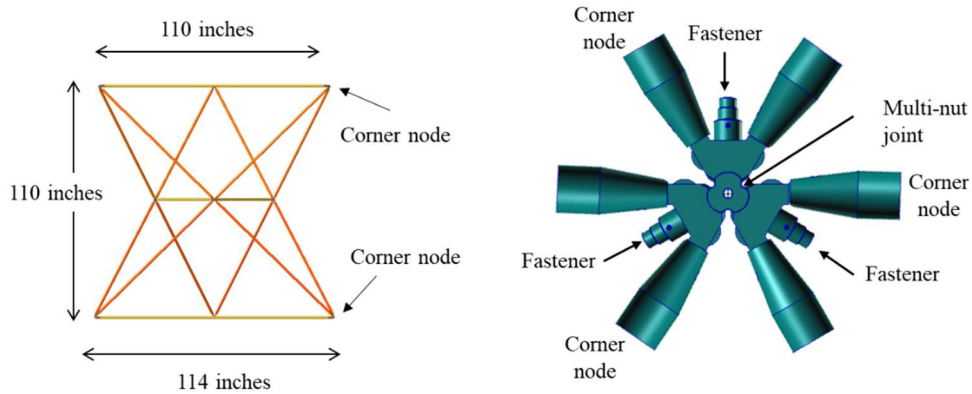


**Fig. 2 The wedge-shaped TriTruss module and multi-nut joint.**

In the initial trade study [4], the governing equations for mass, dimensions, and frequency of a multi-ring telescope support structure were developed. The governing equations for the multi-ring telescope support structure were used to perform rapid and realistic parametric trade studies, allowing major drivers in precision reflector designs to be determined. Finite element models (FEMs) of TriTruss modules were developed with general Timoshenko (shear flexible) beam elements. Individual module FEMs were connected by rigid points at the multi-nut locations to construct a multi-ring telescope support structure. Frequency and static analyses were performed using the FEMs to investigate the overall structural response of TriTruss modules throughout the assembly process and to support various tests involving robotic operations. Although the existing governing equations and FEMs utilizing beam elements worked well in facilitating trade studies and providing preliminary structural response for the multi-ring telescope support structure, an improved analytical approach for the multi-nut joint model is essential for a comprehensive analysis of the multi-ring telescope support structure.

Tests were developed to characterize the assembled multi-nut joint's equivalent axial stiffness (EA) and equivalent bending stiffness (EI). The axial test was developed to determine equivalent EA based on the axial test program summarized in Ref. [6]. The bending test plan was developed to determine equivalent EI based on Ref. [6,7]. For prediction and test correlation, FEMs of axial and bending test specimens were developed by solid elements and the interface between connected nodes and multi-nut joint was implemented as a contact boundary condition with friction. Axial and bending tests were performed at LaRC to measure structural response of the axial and bending test specimens, including forces and displacements, and compared with the predictions obtained from FEM. The details of the test setup and procedures used for the axial and bending tests of the multi-nut joint, along with the test results were summarized in Ref. [8].

This paper presents a summary of the axial and bending tests of the multi-nut joint and focuses on the development of the FEM of the bending specimen utilizing general beam elements with EI and EA obtained from tests. The summary of the axial and bending test as well as comparison of selected test results with predicted results of FEM are given in Section III and IV, respectively. The FEM of the bending test specimen with general beam element and the predicted bending test results are presented in Section V. Concluding remarks are presented in Section VI.

### III. Multi-nut Axial Test and Analysis

#### A. Axial test specimen and test setup

The objectives of the axial test of the multi-nut joint were:

- 1) Evaluate the multi-nut joint for axial performance and provide reference data,
- 2) Determine the installation torque value of the attachment fastener to achieve linear load-displacement response in the operation load range,
- 3) Determine an equivalent EA over the two corner nodes and multi-nut joint based on the axial test results,
- 4) Provide multi-nut joint performance results to validate the FEM of a multi-nut joint.

A test specimen with a 7.94-inch-long joint section composed of three corner nodes and a multi-nut joint, was designed for the multi-nut axial test. In a TriTruss module, each corner node consists of two diagonal and two in-plane bonded sections, and each bonded section has a conical and cylinder bonded section. For the axial test specimen, corner nodes were modified to fit in the test fixture by eliminating two diagonal bonding sections. Specially designed thread adapters were bonded to both end cylinder bonding sections of the test specimen. The modified corner node with the threaded adapter is shown in Fig. 3a. The diameter of the multi-nut joint is 1.15 inch, and the height is 0.71 inch with 10-degree taper angle. Three fastener holes in the multi-nut joint are placed 120 degrees apart and aligned with a fastener hole in each corner node. The geometry detail of the multi-nut joint is shown in Fig. 3b. Three modified corner nodes were connected to a multi-nut joint by M6 fasteners (1.77-inch-long steel socket head screws). The test specimen was inserted into female clevis. The test specimen and the axial test setup are shown in Fig. 4.

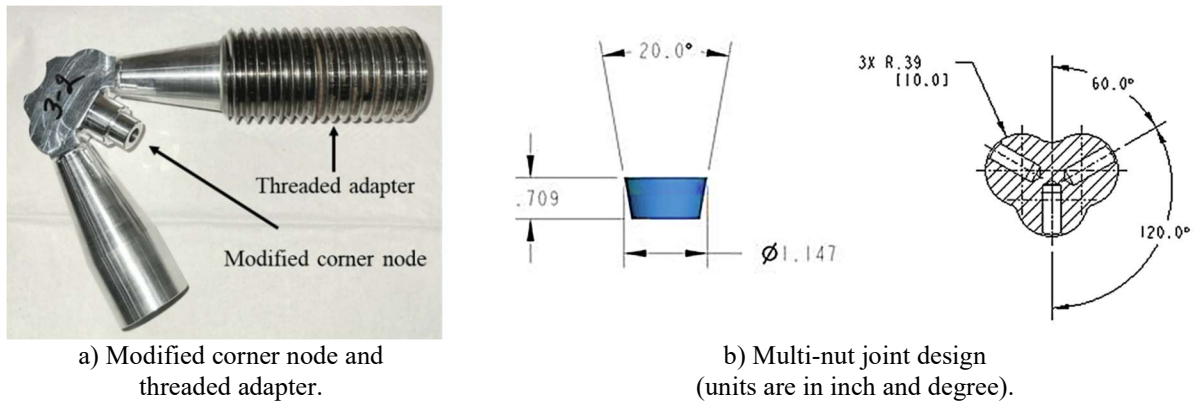


**Fig 3. Corner node and multi-nut.**

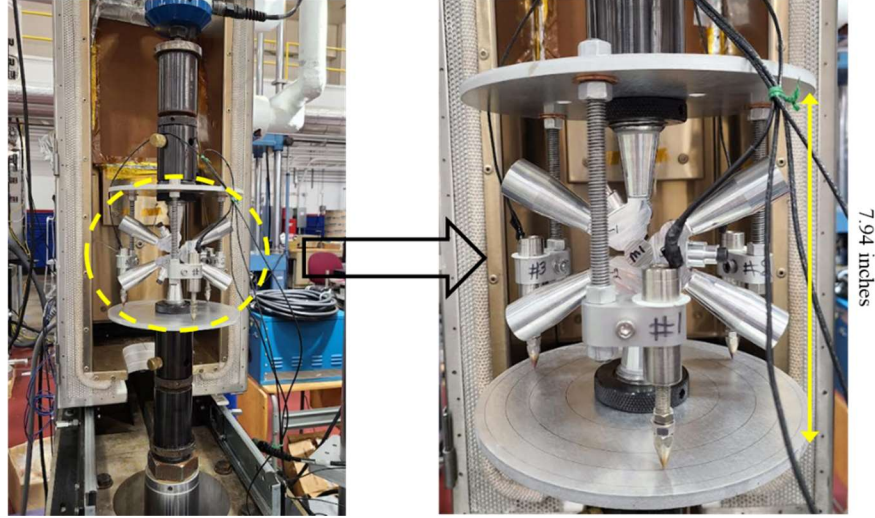


**Fig. 4 Multi-nut test specimen and test setup.**

### **B. Multi-nut axial test and finite element model results**

Five axial test specimens were fabricated utilizing a multi-nut and modified corner nodes. The M6 fasteners with the 1.77-inch and 2.17-inch-long were tightened with four different preload torque values (15 in-lb., 30 in-lb., 60 in-lb., 120 in-lb.). Axial tests were performed on test specimens subjected to maximum tension and compression axial loads of 500 lb.

For test-analysis comparisons, FEMs of the axial test specimen and test fixture were developed using Abaqus<sup>3</sup> eight-node linear brick elements with reduced integration (C3D8R) and ten-node tetrahedral (C3D10) elements [9]. The total number of nodes is 67,639 and total number of elements is 40,980. The isotropic modulus of elasticity of 30 Msi and Poisson ratio of 0.3 were used for the FEM. Fasteners were modeled with two Abaqus general Timoshenko beam elements (B31). Both ends of fasteners were attached using kinematic constraints to multi-nut and corner nodes and the preload torque was applied to the middle node of the beam elements. Contact constraints were applied to the interfaces between the multi-nut and corner nodes. The FEMs of the multi-nut, corner nodes, and fasteners are shown in Fig. 5. The detailed information of test fixture boundary conditions and other analysis approaches of FEM are provided in Ref. [9].

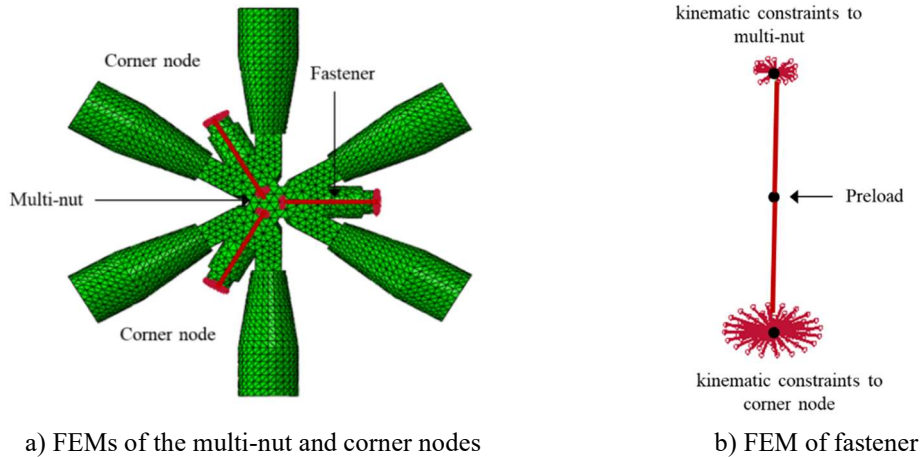


**Fig. 5 Finite element model of multi-nut, corner nodes, and fasteners.**

<sup>3</sup> The use of trademarks or names of manufacturers in this report is for accurate reporting and does not constitute an official endorsement, either expressed or implied, of such products or manufacturers by the National Aeronautics and Space Administration.

The predicted load-displacement responses of the FEM with 1.77-inch-long fastener and a preload torque of 60 in-lb. are shown with test results in Fig. 6. The other test and analysis results were presented in Ref. [8]. In Fig. 6, test results from different test specimens are represented with solid color lines and the predicted results from FEM are represented with the black dashed line. A comparison between the predicted load-displacement response from the FEM and the test results revealed that when the preload torque was greater than 60 in-lb., the predicted response of the FEM consistently matched the test results.

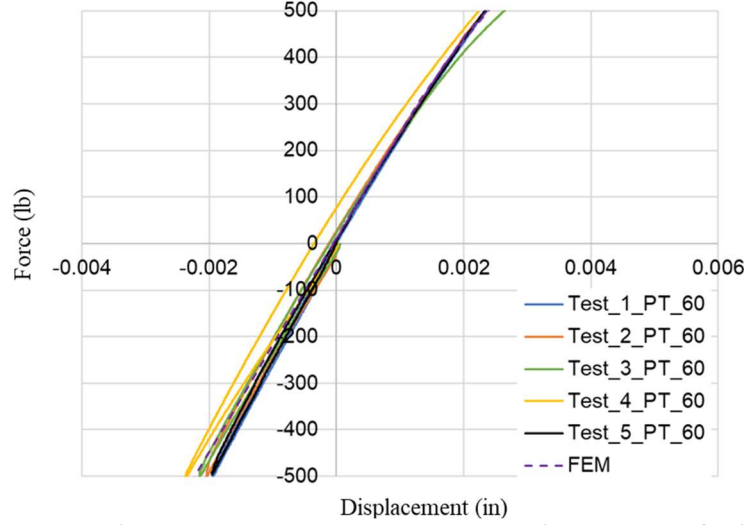


**Fig. 6 The test results and predicted load versus average measured displacement of axial test specimens with preload torque = 60 in-lb.**

### C. Equivalent axial stiffness of multi-nut joint and axial test summary

Equivalent axial stiffness (EA) of the multi-nut joint was calculated to be  $1.697 \times 10^6$  lb. based on the average test result slope ( $EA/L = \Delta P/\Delta d$ ) of the specimen load-displacement response curve, as shown in Fig. 7.

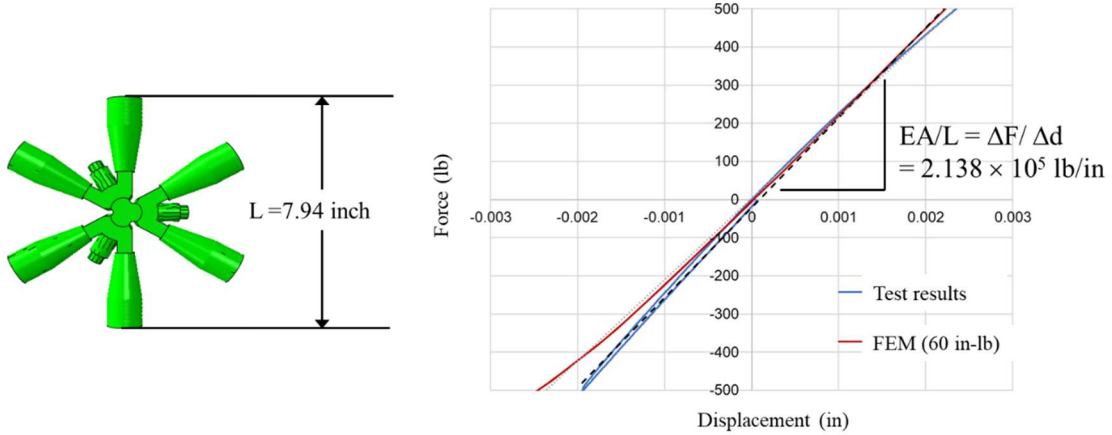


**Fig. 7. Axial stiffness of multi-nut joint.**

The summary of test and analysis results of the axial test include following:

1. The fastener torque and the length of the fastener affected the linearity and repeatability of the joint axial load,
2. Increasing the fastener torque to the specifications increased the tension linearity and repeatability in the load range,
3. Increasing the length of fastener also improved repeatability and linearity by engaging more threads,
4. The preferred installation preload fastener torque value and the length of fastener were found to be 60 in-lb. and 2.17-inches, respectively.

## IV. Multi-nut Joint Bending Test and Analysis

### A. Bending test specimen and test setup

The multi-nut joint bending tests were conducted utilizing a three-point bend test setup, similar to the one described in Ref. [6]. The objectives of the bending test were to:

1. Evaluate multi-nut joint performance bending stiffness performance and provide reference data,
2. Determine an equivalent joint equivalent bending stiffness (EI) of the multi-nut joint over the two corner nodes and multi-nut joint for different joint orientations,
3. Provide multi-nut joint bending performance results to validate the FEM of multi-nut joint.

Three 6-foot-long test specimens were fabricated for the multi-nut bending test. These specimens included a 7.94-inch-long joint section comprising three corner nodes and a multi-nut joint as shown in Fig. 8.

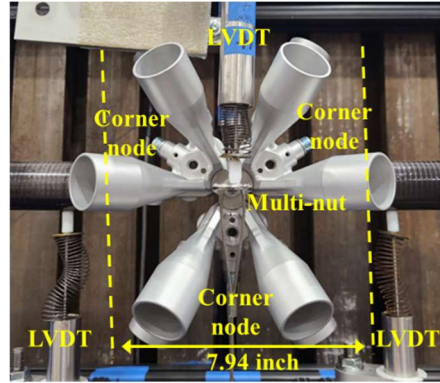


Fig. 8. 7.94-inch-long joint section.

The multi-nut joint was evaluated for two preload values (60 in-lb. and 120 in-lb.). The test specimens were subjected to four different bending orientations as shown in Figure 9: downward, upward, side 1, and side 2 orientations. The multi-nut joint section with these four test orientations respect to global test coordinate is shown in Fig. 9.

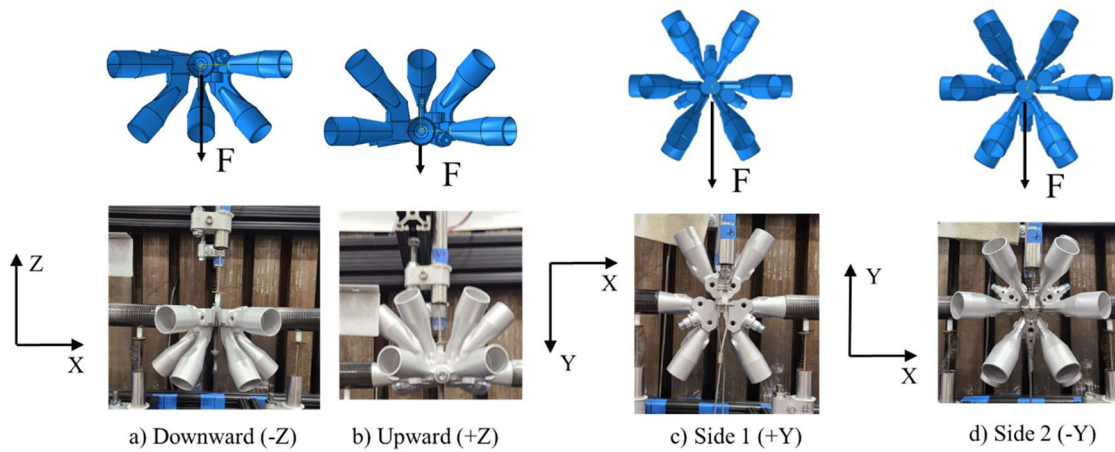


Fig. 9. Bending test orientations.

### B. Multi-nut bending test and finite element model results

For each test specimen, the load was applied to the test specimen until the deflection approached 1 inch. The mid-point deflection of the test specimen was measured by Linear Variable Differential Transformer (LVDT), and test results were processed as load-deflection responses of the test specimens with preload torques of 60 in-lb. and 120 in-lb. for different loading orientations.

For test-analysis comparisons, a FEM of the bending test specimen was developed using Abaqus eight-node linear brick elements with reduced integration (C3D8R) and ten-node tetrahedral elements (C3D10) elements [9]. Fasteners were modeled with two Abaqus general Timoshenko beam elements (B31). Both ends of the fasteners were attached via kinematic constraints to the multi-nut and the corner node and the preloaded torque was applied to the middle-connected node of beam elements. The contact constraints were applied to the interfaces between the multi-nut and corner nodes. Fastener modeling with contact conditions and beam elements are presented in Fig.10.

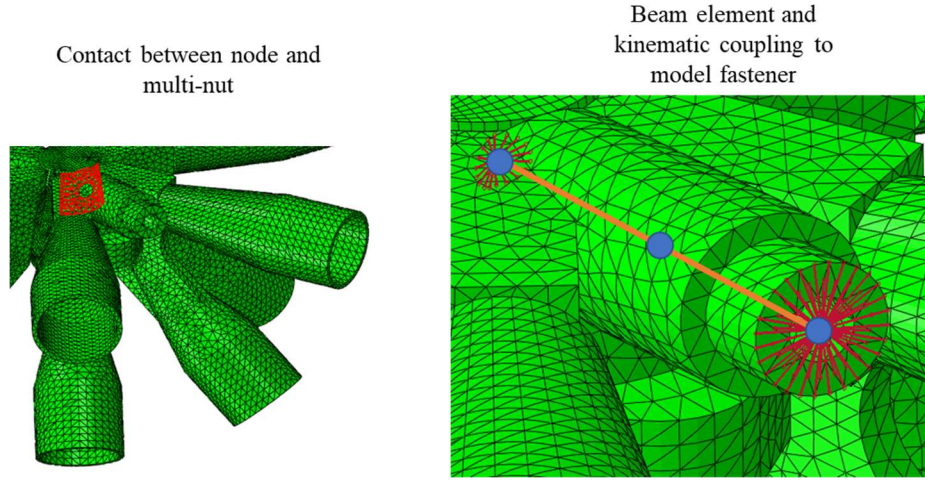


**Fig. 10. Fastener modeling with contact condition and beam elements.**

Four different bending orientations were applied to the FEM of the bending test specimen using different displacement constraint conditions at the simply supported locations. Boundary and loading conditions of the FEM for the bending test specimen for downward and side 2 orientations are shown in Fig. 11.

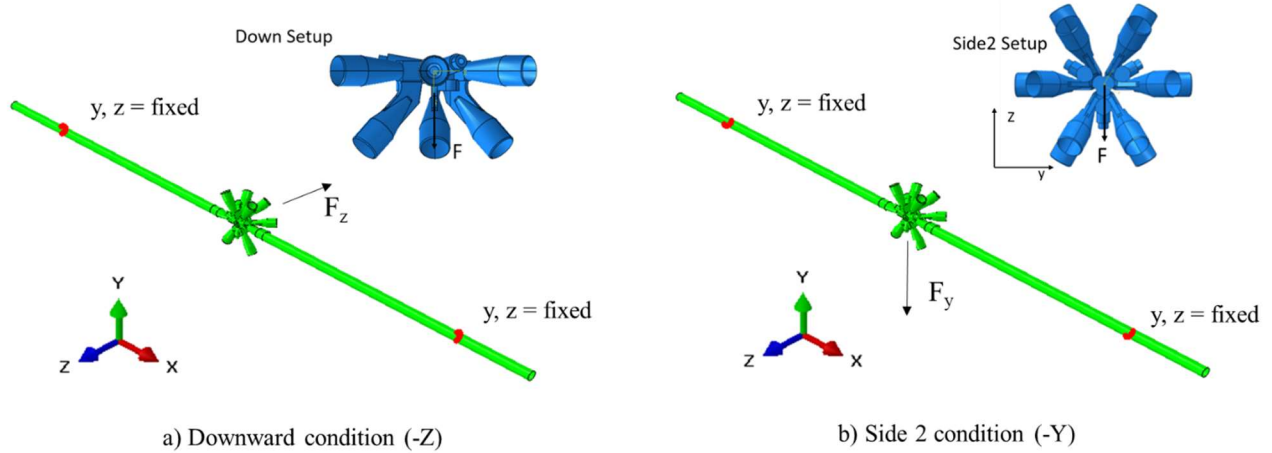


**Fig. 11. Boundary condition of finite element model of the bending test specimen.**

The summary of test-analysis comparison results of the load-deflection responses for the test specimens in four different bending orientations are presented in Fig. 12. Other test results and detailed test-analysis comparison were provided in Ref. [8]. In Fig. 12, the bending response of the test specimen in XZ plane is represented by the load-deflection responses of the downward and upward orientations, and the bending response of the test specimen in XY plane is represented by the load-deflection responses of the side 1 and side 2 orientations. Test results are represented by the solid color lines, and the predicted results from FEM are represented by the dashed lines.

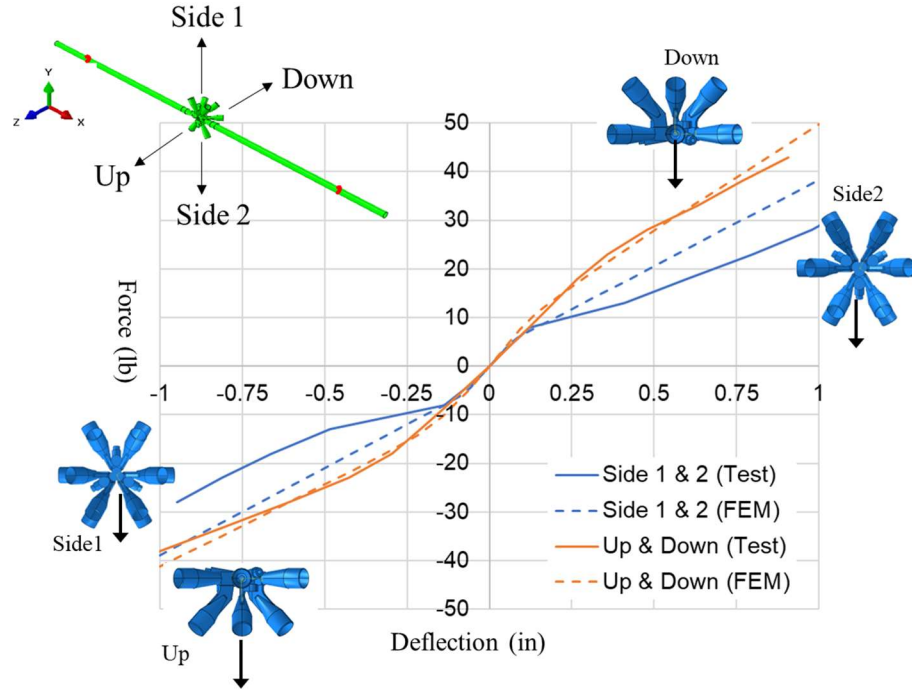


**Fig. 12. Test results and predicted load-deflection responses for four different orientations.**

For all four different bending orientations, test results exhibited an initial linear load-deflection response. As the load was increased, existing gaps between the multi-nut and fasteners, as well as the interface between corner nodes and the multi-nut, closed and interlocked. Consequently, the load-deflection responses of the test specimens changed from the initial linear slope to a nonlinear behavior as the physical gaps were closed. For the side 1 and side 2 orientations, test results exhibited earlier transition of nonlinear load-deflection response from the initial linear response compared to the downward and upward orientations due to complex geometry and initial assembly conditions. The predicted load-deflection responses of FEMs under downward and upward orientations are consistent with test results. The FEM was able to simulate the initial linear and nonlinear load-deflection response consistent with test results. For side 1 and side 2 orientations, the linear load-deflection responses of the test specimens were predicted using FEMs. Test-analysis comparisons indicated that FEMs of the test specimen were not adequate to predict the load-deflection response consistent with test results due to lack of modeling details around the interface between fasteners, the multi-nut, and nodes. Fasteners in the multi-nut joints were modeled with beam elements but the clearance between fasteners and the multi-nut was not included in the FEM. The complex geometry in the interface between corner nodes and multi-nut surfaces was also not captured in FEMs with sufficient fidelity.

Sensitivity analyses of the friction on the interface between the corner node and multi-nut joint were performed to add interaction between nodes and the multi-nut joint. For the sensitivity analysis, three different friction parameters were added in the contact constraint between the nodes and multi-nut surfaces. Test-analysis results of load-deflection response of the test specimens with preloaded torque of 60 in-lb. and 120 in-lb. for side 1 orientation are shown in Fig. 13 and 14, respectively. In Fig. 13 and 14, test results are represented with solid lines while the predicted load-deflection responses from FEMs with different friction parameters are represented with dashed lines. The predicted load-deflection responses from FEMs with friction parameters were characterized by initially linear response and the interlocked condition as the load was applied. An increase of the friction parameter in the FEMs resulted in the interlock condition occurring at a higher applied load. Although FEMs were not adequate to predict the slope of the load-deflection response and the nonlinear response, for both preload torque values of 60 in-lb. and 120 in-lb., and the friction parameter of 0.15, the FEM simulations were adequate to capture the interlock condition and produce load-deflection response similar to the test results.

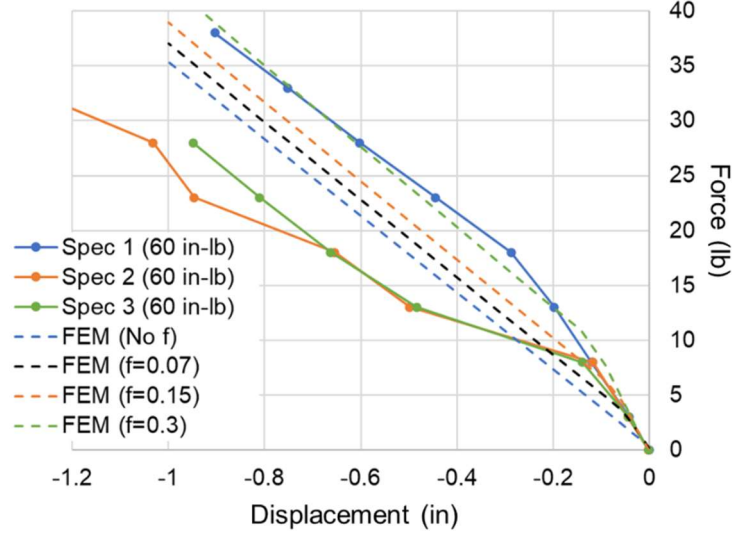


**Fig. 13. Test and analysis results of FEMs with different friction parameter for side 1 orientation (Preload torque = 60 in-lb.).**

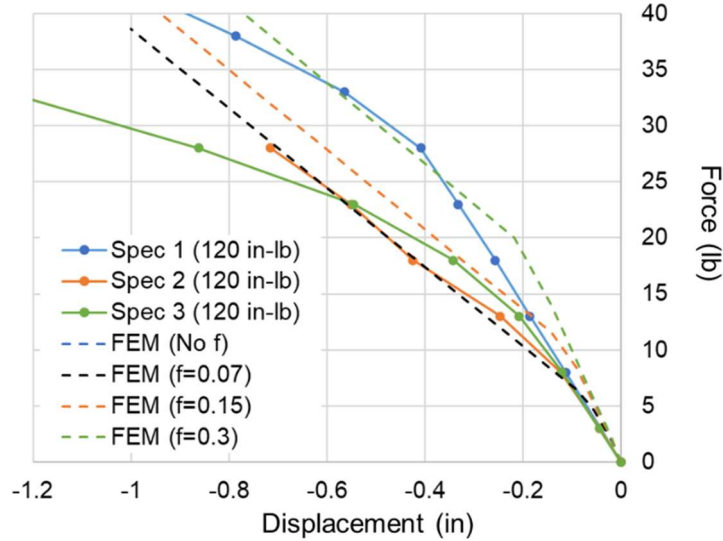


**Fig. 14. Test and analysis results of FEMs with different friction parameter for side 1 orientation (Preload torque = 120 in-lb.).**

### C. Equivalent bending stiffness of multi-nut joint

The equivalent bending stiffness of the multi-nut joint,  $EI_j$ , with the preload torque of 60 in-lb. was calculated by Equation 1.

$$EI_j = \left( \frac{L^3}{48} - \frac{a^3}{6} \right) \div \left( \frac{y}{P} - \frac{a^3}{6EI_s} \right) \quad (1)$$

where  $L$  is the distance between support,  $a$  is the distance from the support to the joint-node interface,  $y$  is the center deflection, and  $EI_s$  is the equivalent bending stiffness of the composite strut [7].

The equivalent bending stiffness of three multi-nut joints was computed. Then, the average equivalent bending stiffness of three multi-nut joints was normalized by the composite strut bending stiffness and was plotted against the corresponding loads for the bending orientation in Fig. 15. The equivalent bending stiffness of the multi-nut was found to vary significantly for given bending orientations. As shown in Fig. 15, the normalized stiffness is reduced as the load is increased. For side 1 and side 2 configurations, the normalized stiffness reaches an asymptote around the load of 10 lb., while for the up and down configurations the normalized stiffness asymptotes at a slower rate.

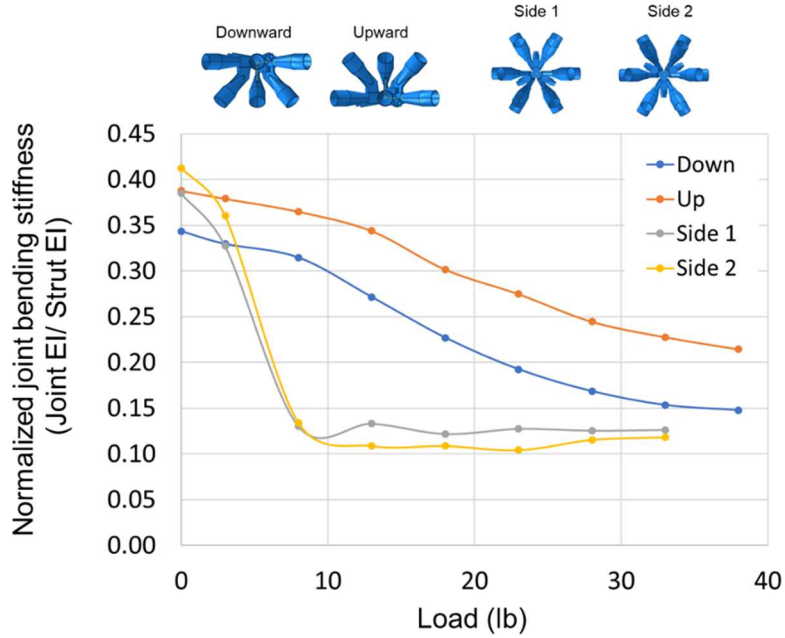


**Fig. 15. Normalized joint equivalent bending stiffness.**

Among the four different bending orientations, the results for the upward orientation show a stiffer equivalent bending stiffness (22% of the bending stiffness of the composite strut with 0.9 mid-deflection). For downward and upward orientations, the response is a nonlinear decrease of the equivalent bending stiffness. This represents the change in interaction between the corner nodes and multi-nut surface from an initial assembly condition to an interlock condition. Equivalent bending stiffness of side 1 and side 2 orientations are characterized by the sudden drop of the equivalent bending stiffness and converge to 12 % of the bending stiffness of the composite strut. The drop of the bending stiffness for side 1 and side 2 orientation is due to slip along the interface between nodes and the multi-nut surface. The minimum equivalent bending stiffness values of the multi-nut joint for different bending orientations are summarized in Table 1. Based on the minimum bending stiffness and the load-deflection response for different bending orientations, the bending behavior of the multi-nut joint could be modeled and analyzed utilizing a torsion spring and other connector elements.

**Table 1. Minimum equivalent bending stiffness of multi-nut joint in different bending orientation.**

| Orientation | Bending Stiffness (lb-in <sup>2</sup> ) |
|-------------|-----------------------------------------|
| Downward    | $7.303 \times 10^4$                     |
| Upward      | $1.059 \times 10^4$                     |
| Side 1      | $6.237 \times 10^4$                     |
| Side 2      | $5.845 \times 10^4$                     |

#### **D. Bending test summary**

The bending tests were conducted to measure the mid-deflection of the test specimen under various bending load orientations. The EI of the multi-nut joint was calculated based on the load-deflection test results. For all four different bending orientations, the load-deflection response of the multi-nut joint remained linear from the initial condition until the interlock between fastener and corner node was reached. However, as the bending load increased beyond the interlock condition, a nonlinear response was observed due to the complex geometry of the interlock condition at the node and multi-nut interface. In the case of side 1 and side 2 bending orientations, there was slippage between the multi-nut and nodes before reaching the interlock condition. These results also suggest that increasing the preloaded torque in the multi-nut joint increases the bending linearity and affects nonlinear response of the multi-nut joint.

Based on the observations from the bending test and the calculated minimum bending stiffness for different bending orientations, further modeling efforts are needed to simulate the bending response of the multi-nut joints. This

may involve the incorporation of other connector elements such as gap and torsional spring elements to accurately capture the complex bending behavior of the multi-nut joint.

## V. Multi-nut Joint Bending Analysis with General Beam Element Model

For test-analysis comparisons, three different FEMs (Type A, B, and C) of the bending test specimen were developed using Abaqus' general Timoshenko beam elements (B31) and two-node connector elements (CONN3D2) [9]. All FEMs had the same two 32-inch-long composite tube sections, modeled with B31 elements using the circular pipe section definition and composite material properties. However, the 7.94-inch-long middle joint section, shown in Fig. 8, was modeled differently. This middle joint section consisted of three components: 1-inch-long bonded sections at both ends, a 2.47-inch-long aluminum section, and a 1.0-inch multi-nut.

First, the Type A model had the joint section which was modeled with B31 elements and different beam definitions. Both 1-inch-long bonded sections were modeled with a 0.598-inch radius pipe cross-section and aluminum and overlapped composite properties. The 2.47-inch-long aluminum node sections were modeled with a 0.563-inch radius pipe cross-section and aluminum properties, and the 1-inch multi-nut was modeled with a 0.5-inch radius circular section and stainless-steel properties. Next, the Type B model had the 7.94-inch-long joint section which was modeled with B31 elements having general beam properties. For the Type B model, each B31 element had equivalent  $E$ ,  $I$ ,  $G$ , and  $A$  values based on the test data rather than beam-specific cross section and material definitions. Lastly, the Type C model had B31 elements with general beam properties and the CONN3D2 connector at the center of the joint section. For the Type C model, a connector element was used to simulate the load-deflection response of bending test specimen in the side 1 and side 2 orientations. The connector element was 0.0001-inch-long and consistent with the slot and cardan angle function [9] to provide independent rotational freedom for the side 1 and 2 orientation at the center of the joint section. A summary of the different joint section models is provided in Fig. 16.

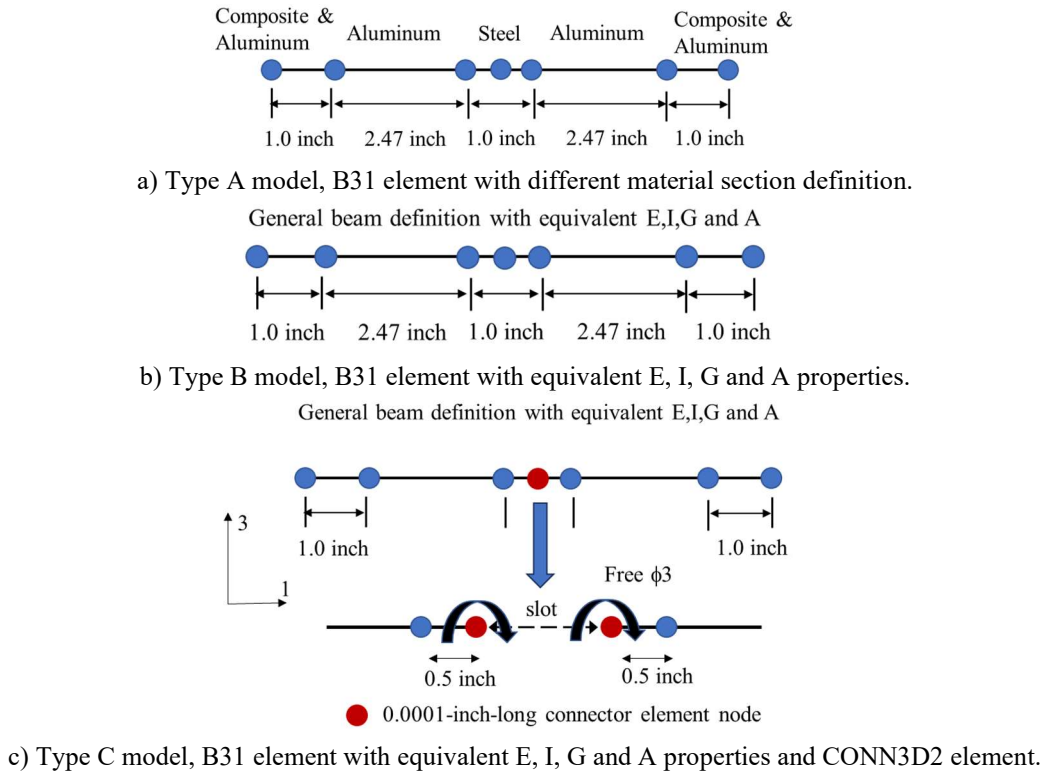


**Fig. 16. Modeling approach of 7.94-inch-long joint section.**

The summary of test-analysis comparison result of the load-deflection responses for the test specimens and the Type A and B FEMs in four different bending orientations is presented in Fig. 17. In Fig. 17, test results are represented by the solid color lines and the predicted results from the Type A and B FEMs are represented by the dashed lines. In Fig. 17, the predicted load-deflection response of the three model types is shown. The cross section of the Type A

model is symmetric for two different bending planes and material properties of aluminum and composite are treated as isotropic [8]. Therefore, the predicted load-deflection responses of the Type A model are consistent for two different bending planes. For the Type B model, although the cross section of the Type B model is symmetric for two different bending planes, the different moment of inertia for bending about the different axis are obtained from Table 1. Therefore, the predicted load-deflection responses of the Type B model are different for each axis similar to test results.

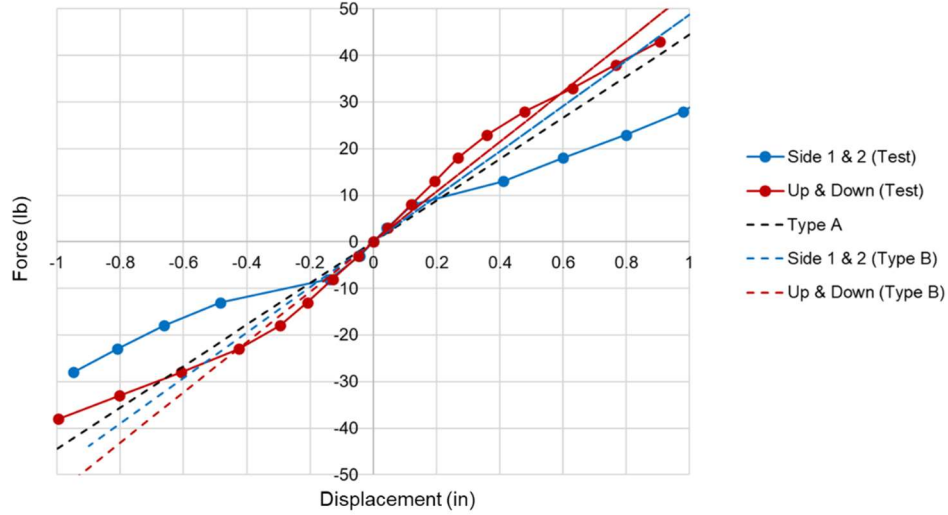


**Fig.17. Test results and predicted load-deflection responses for four different orientations (Beam element models).**

The test-analysis comparison results of the load-deflection responses for the test specimens and FEMs for side 1 and 2 bending orientations are presented in Fig. 18. The load-deflection responses from the Type A and B models are consistent with test results in the initial linear load-deflection response range of -0.2 inch to 0.2 inch. However, the predicted responses are not able to represent the load-deflection behavior of the bending test specimen in the full test range of -1.0 inch to 1.0 inch. In contrast, the load-deflection response from the Type C model matches the load-deflection response of the bending test specimen at higher deflection of -1.0 inch to 1.0 inch but does not capture the stiffer response at small deflections.

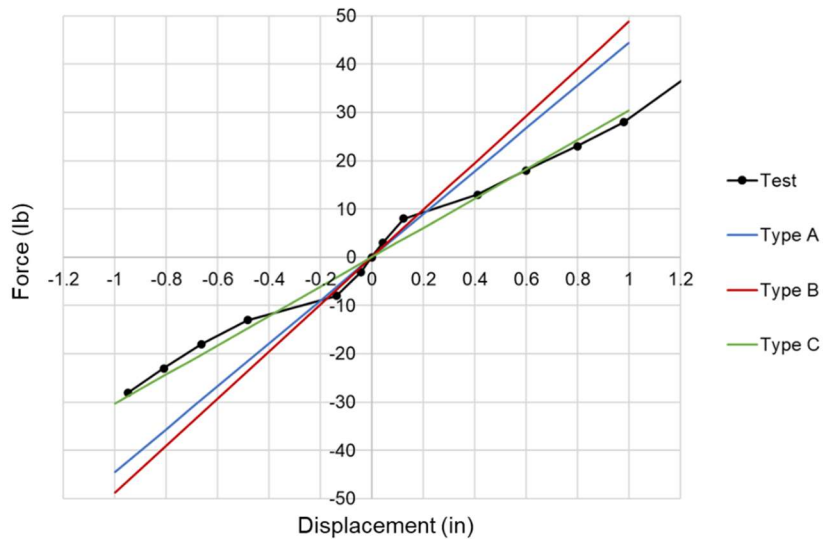


**Fig.18. Test results and predicted load-deflection responses for side 1 and 2 orientations (Beam element models).**

The load-deflection responses of three FEMs of the bending test specimen for up and down orientation represent the linear load-deflection response shown in Fig. 19. The load-deflection responses from the Type A and B models are consistent with test results in the initial linear load-deflection response range of -0.2 inch to 0.2 inch but the predicted responses are not able to represent the load-deflection behavior of the bending test specimen in the full test range of -1.0 inch to 1.0 inch. The load-deflection response from the Type C model is not consistent with test data in the initial linear load-deflection response range of -0.2 inch to 0.2 inch and do not show the transition from the linear to nonlinear response. However, the load-deflection response from the Type C model matches the average slope of the load-deflection curve in the full test range of -1.0 inch to 1.0 inch. In addition, the load-deflection response from the Type C model shows the better load-deflection response compared to the predicted results from the Type A and B models.

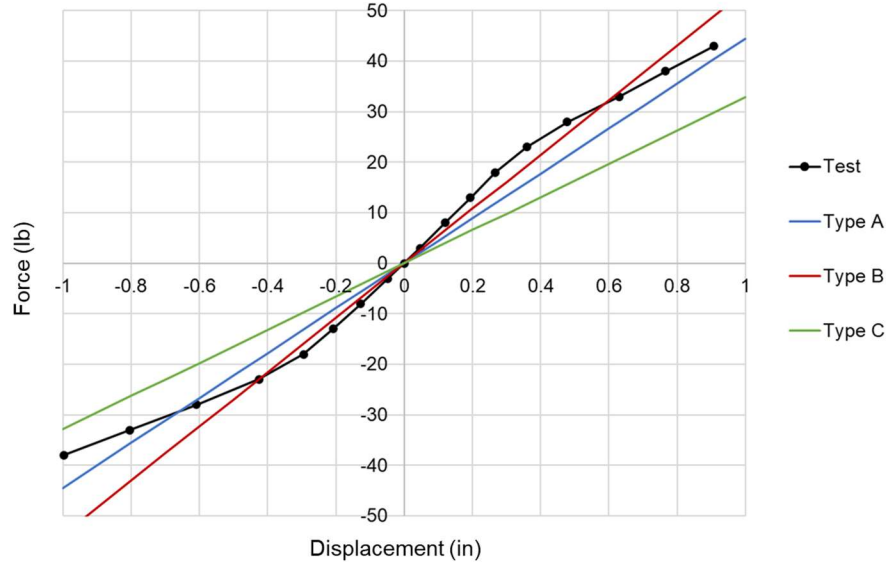


**Fig.19. Test results and predicted load-deflection responses for up and down orientations (Beam Element Models).**

The test-analysis comparison results of the load-deflection responses for the test specimens show that FEMs with the equivalent properties can represent the different bending responses for each bending plane. However, for the full test range of -1.0 inch to 1.0-inch deflection, FEMs with general beam elements couldn't accurately predict the load-deflection responses of the test specimen. During the test, complex structural responses of the interface between the multi-nut and corner nodes, such as slip and closing gap conditions, developed due to complex geometries and assembly conditions. When a connector element was added to the FEM, and the FEM with general beam elements and the connector element was able to predict the load-deflection responses of the test specimen.

## VI. Concluding Remarks

This paper details the axial and bending tests conducted on a multi-nut joint, including the corresponding test and analysis results. The axial tests provide insights into the preloaded torque value required for fastener installation. Additionally, the report presents a sensitivity study on the impact of fastener length and clearance inside the multi-nut on the load-displacement response. The bending tests examine the influence of bending orientation on the load-deflection response of the multi-nut joint. Based on the test results, the axial stiffness (EA) and equivalent bending stiffness (EI) of the multi-nut joint are estimated.

The paper also compares the test results with FEMs for both the axial and bending tests. The comparison shows that the FEMs accurately predict the load-displacement responses of the axial test specimens. For the bending tests, the FEM is modified to predict the bending response of the test specimen for different bending orientations because of interactions of the contact surface between corner node and multi-nut. Different modeling approaches of the bending test specimen with multi-nut joint were discussed which included a FEM with general beam elements and connector elements. The FEMs incorporating both general beam and connector elements successfully predict the load-deflection responses of the bending test specimens.

## References

- [1] Cline, J., Simmons, L., Song, K., White, B., McGlothlin, G., Dorsey, J.T. and Doggett, W.R., “TriTruss Packaging and Deployment Trade Study,” 2020 AIAA Ascend Conference, AIAA 2020-4128, virtual, 16-18 November 2020.
- [2] Doggett, W.R., Dorsey, J.T., Jones, T., Mikulas, M., Teter, J.E., and Paddock, D. “TriTruss: A New and Novel Structural Concept Enabling Modular Space Telescopes and Space Platforms,” 70th International Astronautical Congress, Washington DC, 21-25 October 2019.
- [3] Mukherjee, R., Siegler, N., and Thronson, H., “The Future of Space Astronomy will be Built: Results from the In-Space Astronomical Telescope (iSAT) Assembly Design Study,” 70th International Astronautical Congress, Washington DC, 21-25 October 2019.
- [4] Song, K., Simmons, L., and Cline, J.E., “Modeling Effort of TriTruss Modular Structure for In-Space Assembled Telescope Foundational Structure,” AIAA SciTech 2022 Forum, AIAA Paper 2022-0844, San Diego, CA & Virtual, 3-7 January 2022.
- [5] White, B.W., Doggett, W.R., Dorsey, J.T., and Song, K., “Tessellation and Numerical Simulation of the iSAT Reflector,” 2020 AIAA Ascend Conference, AIAA 2020-4192, virtual, 16-18 November 2020.
- [6] Dorsey, J.T., and Watson, T.T., “Summary of LaRC 2-inch Erectable Joint Hardware Heritage Test Data,” NASA TM-2016-219189, March 2016.
- [7] Wu, K.C., “Characterization of the Bending Stiffness of Large Space Structure Joints,” NASA TM-101565, May 1989.
- [8] Song, K., Stanfield, C., Watson, J., and Stohlman, O., “Structural Analysis and Testing of Multi-nut Joint for TriTruss Structure,” NASA TM-2024-20240002776, March 2024.
- [9] Anon., ABAQUS™ 2021 Analysis User's Guide, Vols. I-V, Version 2021. Dassault Systèmes, Providence, RI, 2021.