Development of a Computational Framework for the Design of Resilient Space Structures

Adnan Shahriar[1]; Sterling Reynolds[2]; Mehdi Najarian[3]; and Arturo Montoya, Ph.D.[4]

[1]Graduate Research Assistant, Dept. of Mechanical Engineering, Univ. of Texas at San Antonio, San Antonio, TX. Email: adnan.shahriar@utsa.edu
[2]Undergraduate Research Assistant, Dept. of Mechanical Engineering, Univ. of Texas at San Antonio, San Antonio, TX. Email: icu327@my.utsa.edu
[3]Graduate Research Assistant, Dept. of Civil and Environmental Engineering, Univ. of Texas at San Antonio, San Antonio, TX. Email: Mehdi.Najarian@utsa.edu
[4]Associate Professor, Dept. of Civil and Environmental Engineering and Mechanical Engineering, Univ. of Texas at San Antonio, San Antonio, TX. Email: Arturo.Montoya@utsa.edu

ABSTRACT

Cyber-physical testing provides a unique platform to enable the design of resilient space structures. This hybrid approach requires the development of a structural model that accounts for various hazards (e.g., micrometeorite and debris impact) and interacts with physical tests and other sub-system models (e.g., thermal) of the space habitat. A two-dimensional finite element analysis code was developed in MATLAB to facilitate the evaluation of potential designs under operating and unexpected loads and prepare the computational framework for eventually performing cyber-physical testing. The code's efficiency was enhanced by using an object-oriented programming approach that reduced data transfer between functions. In this study, the code is implemented to predict the response of a dome-style structure made of regolith concrete to impact loading and identify the force magnitude that will cause the tensile strength to be exceeded in domes with different thicknesses.

INTRODUCTION

The design of space structures in Moon and Mars is a challenging engineering task due to the extreme environmental conditions in space and the complexity involved in evaluating the structural integrity of the proposed designs. Space structures will be exposed to severe extreme temperatures, radiation exposure, micrometeorite and debri impact, reduced gravity, moonquakes, and other unforeseen events. Traditional structural analysis techniques, both analytical and numerical, can be leveraged in the design of space structures. However, these techniques have been shaped through historic structural failures that have provided a deep understanding of conventional loadings in earth (e.g. floods, earthquakes, wind) and revealed potential failure mechanisms caused by these loadings. Hence, the design of space structures pushes the boundary of conventional structural design and requires the use of novel techniques that permit designers to obtain a deeper understanding of the behavior of such unprecedented structures prior to its construction.

Past research efforts towards deep space habitation have focused on performing numerical analyses on different design concepts. For example, Toth and Bagi (2011) proposed a lunar base structure consisting of a masonry vault constructed in a long, narrow valley and within solid rock walls. Malla and Gionet (2013) designed a lunar structure consisting of a Kevlar membrane enclosed by aluminum frames. Mottaghi and Benaroya, (2015) presented the design of an iglooshaped magnesium structure sitting on a sintered regolith foundation. All these studies emphasized on the importance of using in-situ material and considered the use of a thick regolith layer cover to provide protection against radiation and meteorite impact. Nonetheless, the accuracy of the numerical analyses cannot be assessed due to the cost and resources associated with performing experimental tests that replicate deep space conditions.

Ongoing research work by the Resilient Extraterrestrial Habitats Institute (RETHi) will merge numerical analyses and physical tests to perform trade studies that allow RETHi researchers to consider different structural configurations for the habitat structure . Through this cyber-physical approach, physical and mechanical parameters that contribute to the stability, redundancy, and reparability of potential habitat structures can be identified. This research endeavor requires the development of physics-based models, including a model that predicts the mechanical response of the structure to various hazards and receives input from physical tests and other sub-system models (e.g., thermal, power, environment) of the space habitat. This paper shows initial work on the development of a flexible finite element analysis code that facilitates the evaluation of potential space structure designs and can be employed in future cyber-physical testing. The adaptability of the current computational development in the evaluation of potential resilient measures is illustrated by identifying the impact load magnitudes that will cause the tensile strength to be exceeded in a dome-style structure made of regolith concrete with different thicknesses.

METHODOLOGY

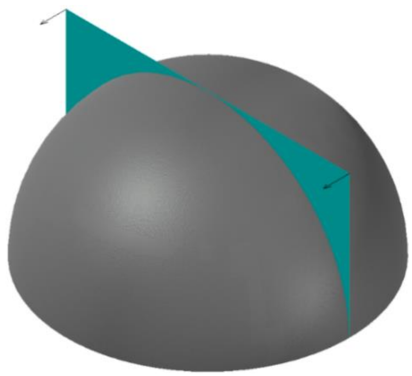
A finite element analysis (FEA) code was developed in MATLAB to predict the displacement, strain, and stress fields of space habitat structures under operating loads (e.g. pressure and temperature fluctuations) and impact loads (e.g. micrometeorite or debri impact). The development of an in-house FEA code was preferred over the adoption of a commercial FEA software, as it provides the advantage of: (a) intruding the code to facilitate is interaction with input parameters from the physical system and other sub-system models of the space habitat, and (b) modifying the code structure to reduce its computational time, which contributes to performing the cyber-physical tests in real time.

At its current stage, the code follows the formulation for a linear elastic material under either static (including thermal stress analysis) or dynamic loading conditions. The model development revolved around the analysis of a monolithic spherical dome structure with an inner radius of 2.5 m as shown in Figure 1. The dome was modeled as a two-dimensional structure under plane strain conditions. The dome base was assumed to be rigidly connected to the lunar surface, meaning all horizontal and vertical displacements beneath the base were prevented (i.e., fixed conditions).

The internal loading consisted of internal pressure applied at the inner surface and gravity load. The loading and geometric parameters are summarized in Table 1.

Table 1. Geometric and Loading parameter

Parameter	Symbol	Value
Internal Radius	R	$2.5\ m$
Wall thickness	w	$0.4\ m$
Internal pressure	P	$101325\ Pa$
Gravity load (acceleration)	G	$1.62\ m/s^2$



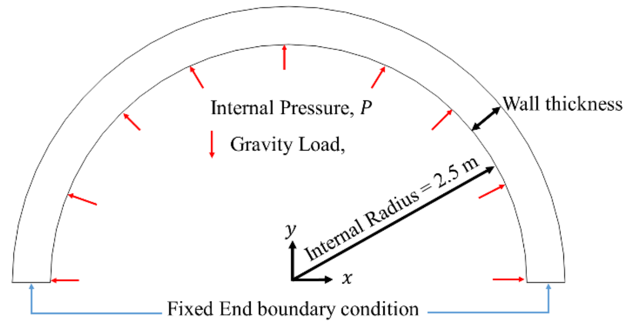


Figure 1. Dome Geometry. 3D view (left) and 2D section view (right).

The dome was assumed to be made of regolith concrete material with the properties provided in Table 2.

Table 2. Material Parameters

Material Property	Symbol	Value
Density	ρ	$1400\ kg/m^3$
Elastic Modulus	E	$14\ GPa$
Poisson's ratio	N	0.3
Linear thermal expansion coefficient	A	$0.000001\ K^{-1}$

These properties correspond to typical values for regular concrete at room temperature, as regolith concrete is expected to have similar properties as regular concrete. However, due to the large temperature fluctuations experienced at extraterrestrial environments, temperature dependent variations of the physical and mechanical properties will be eventually incorporated to the model.

Mesh

The dome model was meshed with 4-noded elements that assume a linear displacement interpolation within the element as shown in Figure 2. The dome was discretized using a structured mesh, which was created by specifying the spacing in degrees between the nodes around the dome's semi-circle and the number of elements through the thickness. The meshing routine developed in MATLAB provides flexibility in adjusting the mesh according to the modeling needs.

Impact

Impacts due to micrometeorite impact or debri were introduced to the model by using a simplified approach proposed by Malla and Gionet (2013). This approach approximates impact by applying a concentrated force at a nodal point of the outer surface of the dome as shown in Figure 3. This impact force varies as a function of time according to an amplitude function. The micrometeorite is characterized by its rigid mass, m_2, and velocity, v_2. For a dome with mass m_1, section length L, and moment of inertia I, the impact force $Q(t)$ can be calculated as,

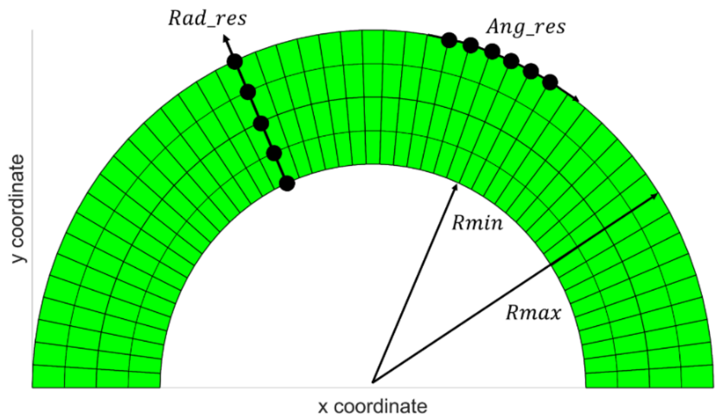


Figure 2. Geometric parameters defining the FEA model of the dome structure.

$$Q(t) = \frac{17}{35} m_1 \ddot{n} + \frac{48EI}{L^3} n \tag{1}$$

where the parameters $n(t)$ and $\ddot{n}\,(t)$ are defined as follows,

$$n(t) = \frac{m_2 g}{k}(1 - \cos(\alpha t)) - \frac{m_2 v_{2,0}}{\sqrt{\beta k}} \sin(\alpha t) \tag{2}$$

$$\ddot{n}(t) = \frac{\alpha^2 m_2 g}{\beta} \cos(\alpha t) - \frac{\alpha^2 m_2 v_{2,0}}{\sqrt{\beta k}} \sin(\alpha t) \tag{3}$$

The constants, k, α, β are

$$k = \frac{48EI}{L^3} \tag{4}$$

$$\alpha = \sqrt{\frac{k}{m_2 + \frac{17}{35} m_1}} \tag{5}$$

$$\beta = m_2 + \frac{17}{35} m_1 \tag{6}$$

The parameters used to determine the impact force magnitude are listed in Table 3.

The resulting sinusoidal type amplitude, calculated according to Equation 1 and the parameters listed in Table 3, is illustrated in Figure 3. The maximum force magnitude is 156 kN. The impact is solved by using a Newmark's implicit integration scheme and follows classical Rayleigh damping assumption, $\mathbf{C} = a_0\mathbf{M} + a_1\mathbf{K}$, where a_0 and a_1 are constants of proportionality associated with the mass, **M**, and stiffness matrix, **K**. More advanced approaches for modeling impact and damping will be included in later developments of the code.

Table 3. Micrometeorite Parameters

Parameter	Symbol and Equation	Value
Micrometeorite mass	m_2	$0.088\ kg$
Micrometeorite velocity	v_2	$100\ m/s$
Section Length	$L = \frac{\pi}{2}(2R + w)$	$8.48\ m$
Moment of inertia	$I = \frac{\pi}{8}[(R + w)^4 - R^4]$	$12.43\ m^4$
Dome mass	$m_1 = \frac{4\pi\rho}{6}[(R + w)^3 - R^3]$	$2.57 \times 10^4\ kg$

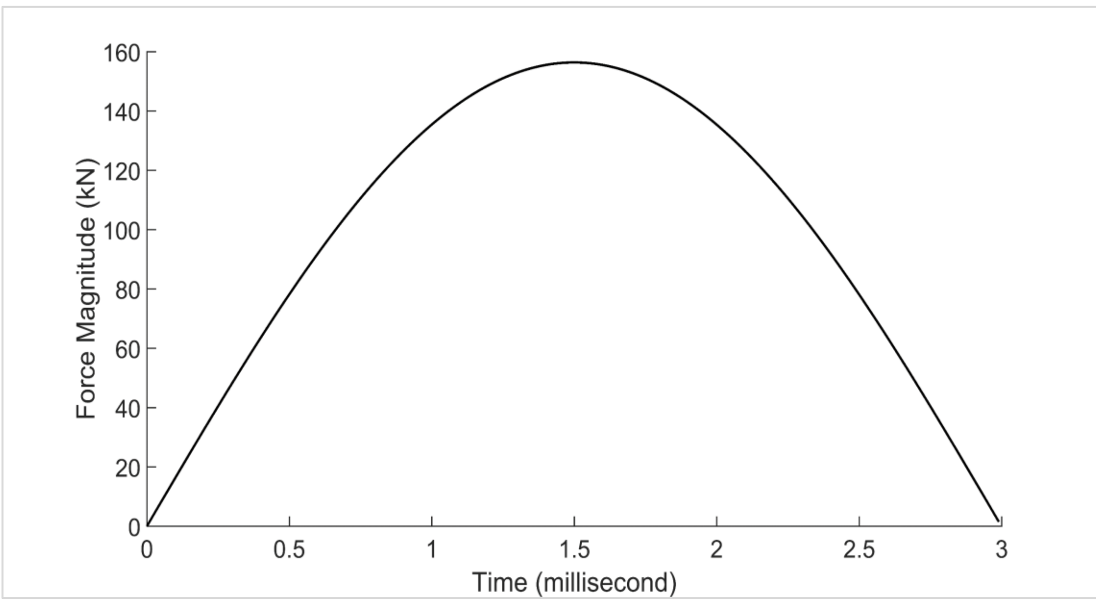


Figure 3. Impact load as a function of time

Code Structure

FEA codes are characterized by having multiple functions, devoted to specific tasks of the FEA solution process. The structure of the current code is provided in Figure 4. Calculating the element stiffness matrix of every element involves using the material properties (e.g. Elastic Modulus) and nodal coordinates that were originally defined in the input file. Feeding inputs directly to a function or using global variables can significantly increase the computational time of the code. Hence, an object-oriented programming approach was used to reduce data transfer between functions. Under this approach, external functions (e.g. assembly function) can retrieve data (e.g. connectivity matrix) from the properties of the class, i.e., a location within the memory storage, in order to expedite the computation of FEA variables (e.g. global stiffness matrix). The code has been verified by comparing its solutions to those provided by commercial software.

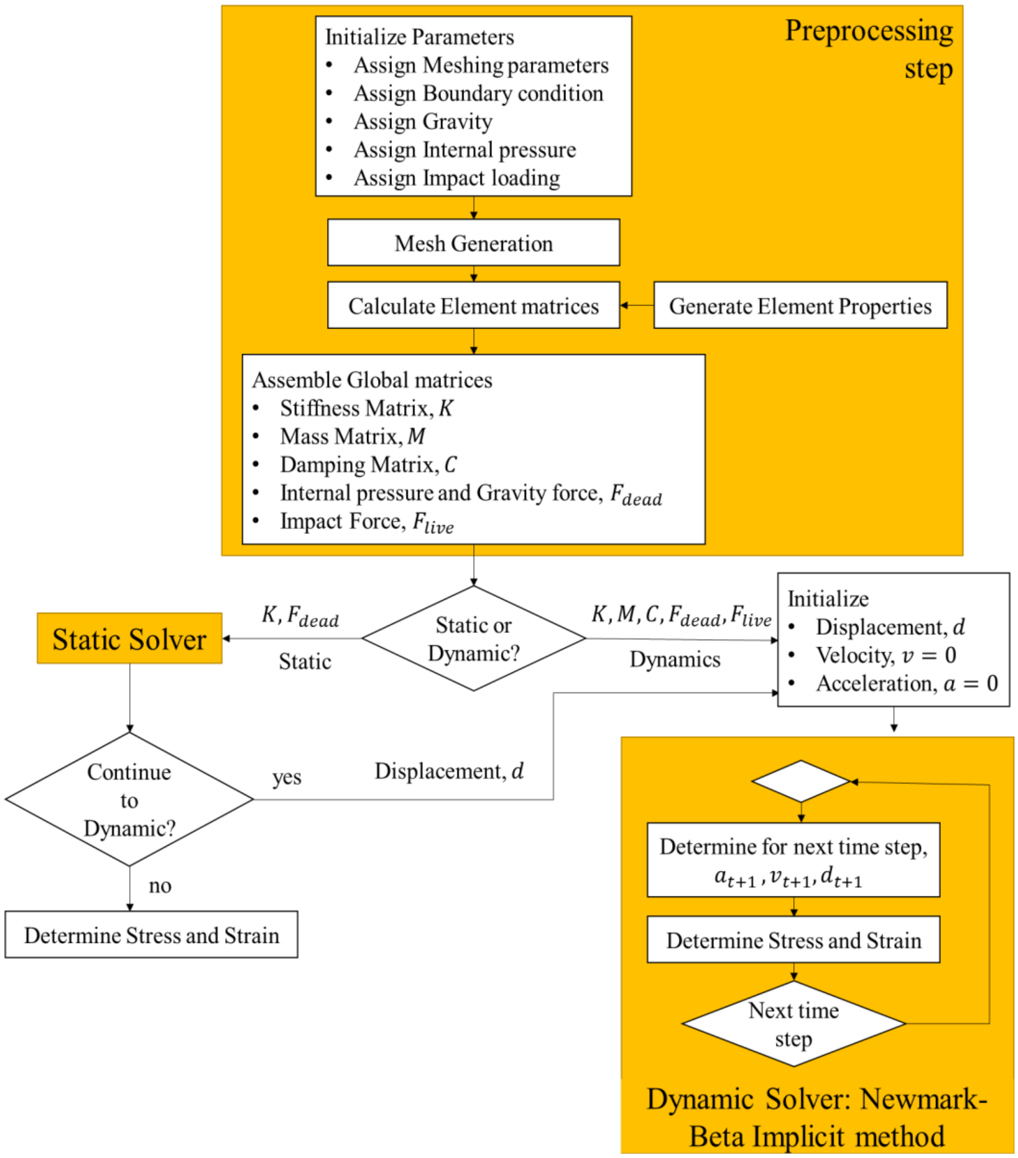


Figure 4. FEA code flowchart

RESULTS

Impact

The hoop stress response for the 0.4 m thick dome due to the impact force indicated in Figure 3 at a 45° angle (see Figure 5) is illustrated in Figure 6 for three different damping coefficient combinations (a_1= a_0=0, 0.001, 0.01). The solution was obtained for the first 0.01 s after the impact in time increments of 0.0001 s. Figure 6 normalizes the hoop stress with respect to an assumed tensile strength value for regolith concrete (1.5 MPA) at the point of interest (shown in Figure 5) throughout the simulation. It can be observed that the hoop stress was clearly exceeded for two cases with the lowest damping coefficient. The compressive strength of concrete was assumed as 15 MPa and was not exceeded at any time within the simulation.

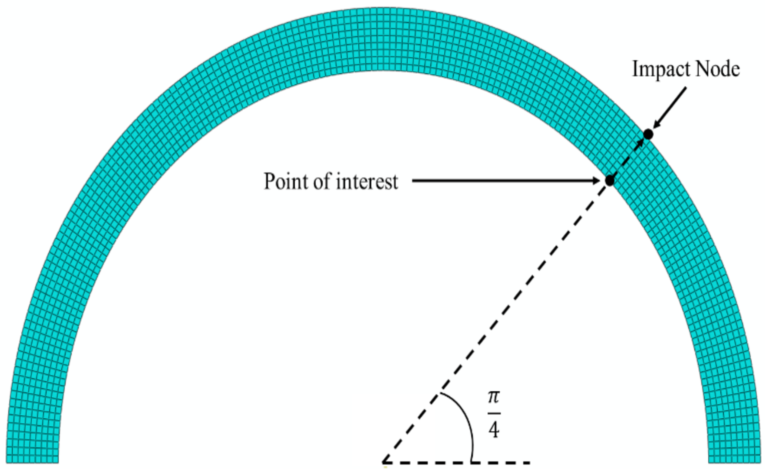


Figure 5. Impact Angle

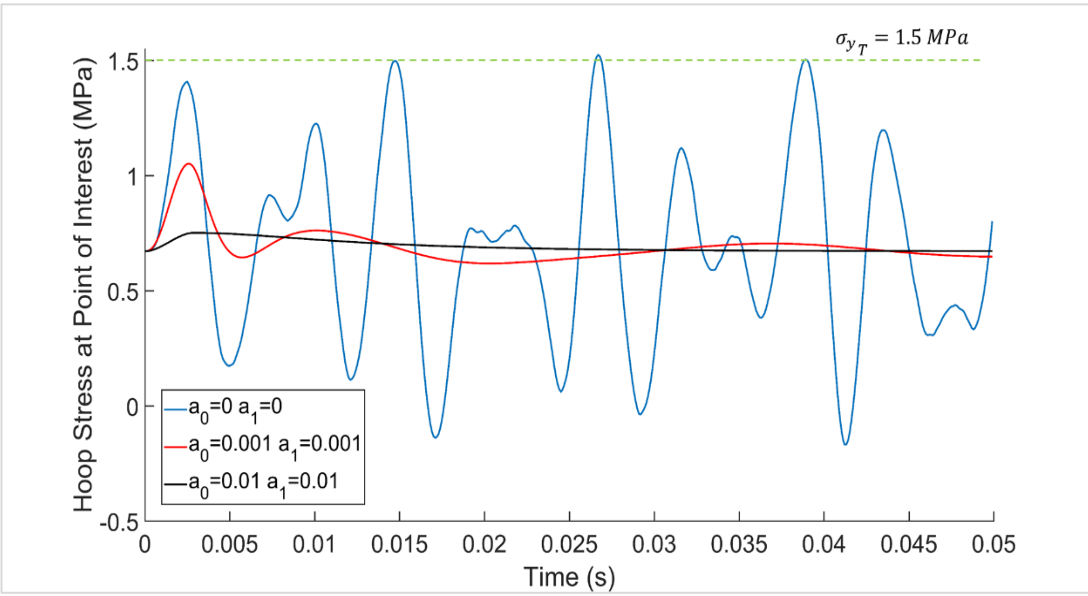


Figure 6. Hoop stress at point of interest shown in Figure 5 for different damping coefficients, where $\mathbf{C} = a_0\mathbf{M} + a_1\mathbf{K}$

Code Efficiency

The code's CPU time as a function of the number of elements employed in the mesh is provided in Figure 7. The internal pressure was applied in a static analysis and then the impact force was applied in an implicit analysis. The solution was obtained for a total of 0.05 s after the impact in increments of 0.0001 s. The computational time increased with number of elements as expected, but the overall computation was kept below 100 s even for the denser mesh (3781 elements), as shown in Figure 7. Nevertheless, further coding alternatives, e.g. parallel computing, are being considered to increase the efficiency of the code and allow an execution as close to real time.

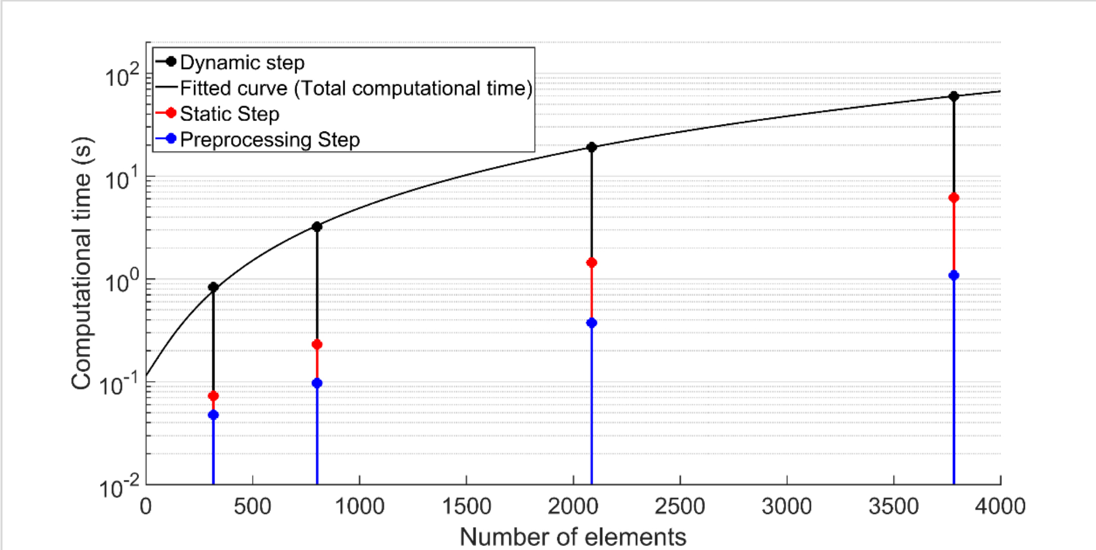


Figure 7. CPU time

Parametric Study

The thickness of the dome plays a crucial role in determining the load magnitude that will cause tensile yielding of the dome. The analysis was repeated in order to identify the load magnitude that caused a section of the dome to exceed the assumed tensile yielding of regolith concrete. The flexibility of the FEA code allowed to easily modify the mesh and loading of each run. The size of the individual elements was kept constant for each run to avoid mesh size effects. Figure 8 indicates that the increased strength in the dome as function of thickness follows a quadratic trend.

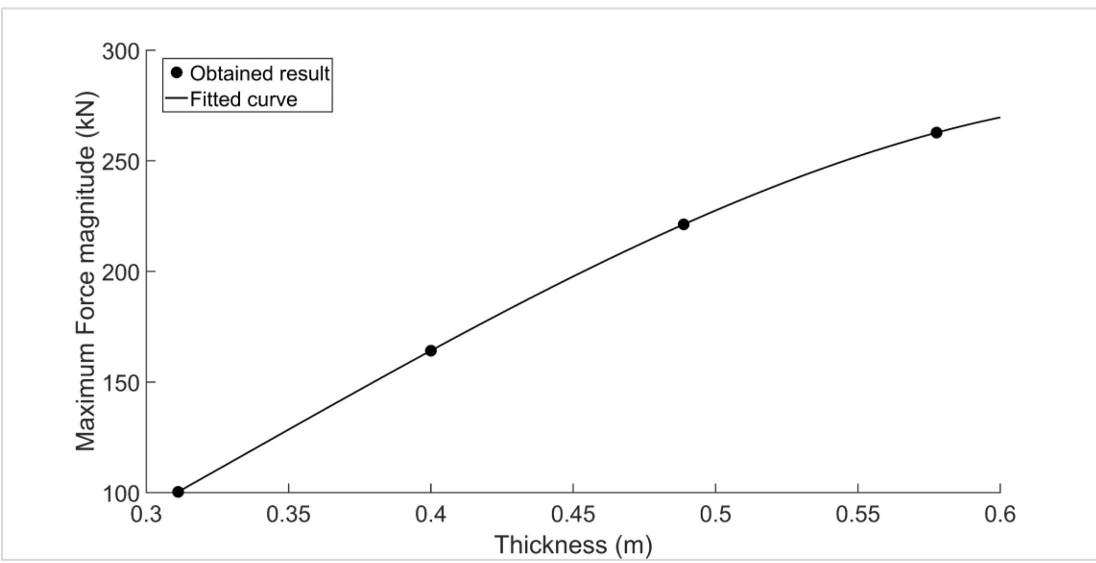


Figure 8. Force Magnitude causing tensile yielding versus Dome Thickness

CONCLUSION

This paper shows initial work on the development of an in-house finite element analysis code that will facilitate the evaluation of different design options for habitat structures and be eventually used for conducting cyber-physical testing. The current stage of the code conducts linear and dynamic analyses in an efficient manner through an object-oriented programming approach. The code was found useful in identifying the load magnitudes that will cause the tensile strength to be exceeded in domes with different thicknesses. The code will be further used to identify physical and mechanical parameters significantly contributing to the structural resiliency of different space habitats.

ACKNOWLEDGMENTS

This work conducted under Resilient Extraterrestrial Habitats Institute (RETHi) was supported by a Space Technology Research Institutes grant (number 80NSSC19K1076) from NASA's Space Technology Research Grants Program. The authors are thankful to Ramesh B. Malla, Ph.D., F. ASCE, Professor at the University of Connecticut and Amin Maghareh, Ph.D., Research Assistant Professor at Purdue University for their valuable inputs to the development of the code.

REFERENCES

Malla, R. B., and Gionet, T. G. (2013). Dynamic response of a pressurized frame-membrane lunar structure with regolith cover subjected to impact load. *Journal of Aerospace Engineering*, *26*(4), 855-873.

MATLAB. (2018). *R2018a*. The MATHWORKS, Inc. Natick, Massachusets, USA.

Mottaghi, S., and Benaroya, H. (2015). Design of a lunar surface structure. I: Design configuration and thermal analysis. *Journal of Aerospace Engineering, 28*(1), 04014052.

RETHi. (n.d.). Retrieved November 22, 2019, from Resilient ExtraTerrestrial Habitats Institute: https://www.purdue.edu/rethi/.

Tóth, A. R., and Bagi, K. (2011). Analysis of a lunar base structure using the discrete-element method. *Journal of Aerospace Engineering, 24*(3), 397-401.