

Modelling and Analysis of Scaffolding Structure Used in Aerospace Vehicle

M. Ganesh¹, K. Shiva Shankar²

¹Department of Aeronautical Engineering, MLR Institute of Technology, Hyderabad, Telangana State, India

²Department of Aeronautical Engineering, MLR Institute of Technology, Hyderabad, Telangana State, India

Abstract: The project elucidates the methodology of design and analysis of scaffolding structure for a payload of a typical flight vehicle. Scaffolding is a temporary structure used to support any material in the construction of buildings and other large structures. The scaffolding structure with munitions is called payload. Payload is an explosive material that is delivered by a flight vehicle. Here the scaffolding structure is used for mounting the munitions. It is usually a modular system of metal pipes or tubes, although it can be from other materials. Generally in the flight vehicles the material used will be Aluminum. Each munition's weighs 27 kg (approx). Scaffolding structure has to be designed for mounting 28 munitions (approx). This payload is mounted in a flight vehicle. The flight vehicle experiences maximum longitudinal acceleration of 4g and a lateral acceleration of 2g. Our project deals with the design and analysis of the arrangement of munitions in the Payload using the scaffolding system. The scaffolding structure need to be designed such that the maximum permissible allowable deflection on the scaffolding structure is 0.5 mm with a minimal factor of safety of 2. Pro Engineer CAD software is used for modeling, ANSYS for linear static analysis. The results are compared with theoretical calculations.

Keywords: scaffolding structure, Pro Engineer CAD software, ANSYS

1. Introduction

Flight/Aerospace Vehicle is a type of transportation device which is capable of traveling in the air/atmosphere or to the outer space. There are a lot of flight vehicles in operation. Each has their own unique characteristics and functions from the other and can be easily distinguished from the others. The importance of these aerospace vehicles is growing for transportation of deadly warhead that can destroy within its range of operation both on surface and outside the atmosphere. Research interest in this field is also moving at a fast pace. The velocities with which these vehicles are travelling are increasing with increase in the load carrying capacities as well. The structural analysis of the flight vehicle becomes important at this stage as a minor deflection in the structure leads to a major variation in destination or mistake in the structure or failure of a component can lead to hazardous consequences.

2. Purpose of Aerospace Vehicle

The main aim of an aerospace vehicle is to carry the warhead to the required destination. Warhead is decided based on Target to be destroyed. For anti tank, shaped charge is used. The difference between rocket and missile is that rocket cannot be guided after the launch, but missile can be guided till the terminal end. Mid course correction is required for chasing moving targets and also to take care of transients during missile flight.

3. Working of Aerospace Vehicle

Aerospace Vehicle is launched using ground equipment called launcher. On triggering, pulses get out from launcher and start activating power supplying units and the gyroscopic unit. The supply voltage of every power source is checked for its required voltage level by the launcher and after

achieving the same, subsequent actions take place sequentially. Launcher throws the missile out and continuously watches the movement of missile from its line of sight. Missile position is indicating by IR lamp source. Launcher compares the position of missile from the line of target and calculates the correction commands and sends it through the wire communication to the missile. Missile takes the commands and decodes the signal into the corresponding pitch and yaw commands and feeds to the appropriate vanes at appropriate duration of time for steering the missile in the direction. Upon hitting the target, the warhead gets actuated to destroy it.

4. Computer-Aided Engineering

Computer-aided engineering (CAE) is the broad usage of computer software to aid in engineering tasks. It includes computer-aided design (CAD), computer-aided analysis (CAA), computer-integrated manufacturing (CIM), computer-aided manufacturing (CAM), material requirements planning (MRP), and computer-aided planning (CAP). Software tools that have been developed to support these activities are considered CAE tools. CAE tools are being used, for example, to analyze the robustness and performance of components and assemblies. The term encompasses simulation, validation, and optimization of products and manufacturing tools. In the future, CAE systems will be major providers of information to help support design teams in decision making.

5. Pro/Engineer

Creo Elements/Pro, a product formerly/mostly known as **Pro/ENGINEER** is a parametric, integrated 3D CAD/CAM/CAE solution created by Parametric Technology Corporation (PTC). Pro/ENGINEER is a 3D solid modeling application used worldwide by Mechanical Engineers to

design, develop, and analyze products. **Pro/ENGINEER** allows engineers to significantly decrease time to market by making development changes and modifications to a design easy to incorporate.

The parametric modeling approach uses parameters, dimensions, features, and relationships to capture intended product behavior and create a recipe which enables design automation and the optimization of design and product development processes. This design approach is used by companies whose product strategy is family-based or platform-driven, where a prescriptive design strategy is fundamental to the success of the design process by embedding engineering constraints and relationships to quickly optimize the design, or where the resulting geometry may be complex or based upon equations. Creo Elements/Pro provides a complete set of design, analysis and manufacturing capabilities on one, integral, scalable platform. These required capabilities include Solid Modeling, Surfacing, Rendering, Data Interoperability, Routed Systems Design, Simulation, Tolerance Analysis, and NC and Tooling Design.

5.1 Engineering Design

Creo Elements/Pro offers a range of tools to enable the generation of a complete digital representation of the product being designed. In addition to the general geometry tools there is also the ability to generate geometry of other integrated design disciplines such as industrial and standard pipe work and complete wiring definitions. Tools are also available to support collaborative development.

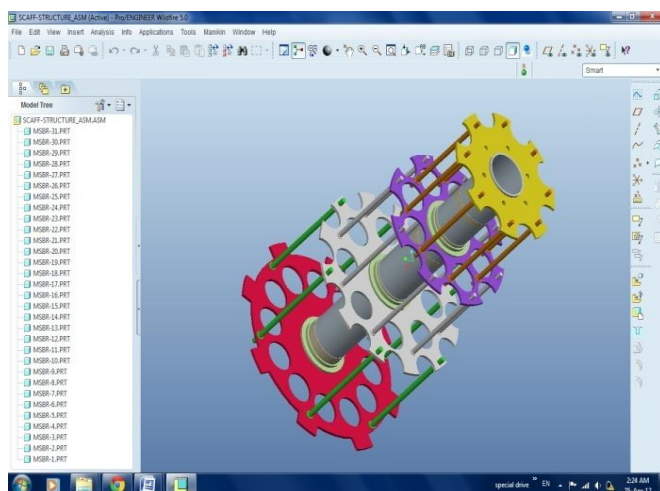


Figure 3: Screen shot captured from Pro/ENGINEER of Scaffolding Structure

A number of concept design tools that provide up-front Industrial Design concepts can then be used in the downstream process of engineering the product. These range from conceptual Industrial design sketches, reverse engineering with point cloud data and comprehensive free-form surface tools.

5.2 Design Specifications

1. Maximum weight of sub munitions = 50 kg
2. High package density (Maximum no of sub munitions

within the given volume to be mounted)

3. Maximum permissible deflection of CST = 0.5 mm
4. Maximum permissible deflection of plate = 1.2 mm
5. Material of CST, Plates – Al. Alloy and Tie rods – EN 24

5.3 Design of Sub Systems

5.4 Design of Support Structure

The support structure of CST and two circular plates mounted on both ends of CST. A middle plate is also mounted in the middle of CST. All the plates are interconnected using tie rods. The complete assembly forms the support structure. the plates are used for assembling the support structure to the flight vehicle airframe through the front and rear frames. The front and rear frames are mounted to the front and rear plates of support structure by using bolts.

Plates will take the total weight of the sub munitions and has to be designed based on stiffness and strength based criteria. The CST, tie rods will prevent the bending of the plates and has to be designed based on stiffness criteria.

The support structure is designed using Theory of Bending. Double integration method for deflection and theory of plates.

5.5 FEM Analysis of Support Structure:

The following are made in the FEM analysis of support structure:

1. The sub munitions weight is divided proportionately between the plates supporting the sub munitions.
2. The front and rear plates are rigidly mounted to the air frame.

5.5.1 Modeling

The support structure is modeled in ANSYS

- The CST is modeled using 10 node 187 element.
- The plates are modeled using Shell element.
- The tie rods are modeled using Beam element.

5.5.2 Central Support Tube

The CST is the prime load bearing member of the war head system as all the sub munitions, ejection mechanisms are mounted on it. The CST is supported at both the ends by war head frames. This warhead makes use of shorter CST vis-à-vis the one used in incendiary sub munitions warhead and also the skew angles are changed to optimize the dispersion pattern and the weight & CG

5.5.3 Design of CST

The CST is analyzed with analytical methods without considering the warhead frames. The CST is considered as simply supported beams carrying number of point loads. The weight of incendiary and P.F sub munitions in row is assumed to be acting as a point load at that row.

During its flight the missile experiences a lateral acceleration of 3g max and longitudinal of 3.5g max. In the analytical

calculation, the effect of only lateral acceleration is considered.

Loading pattern of CST is shown in the sketch below. The shear force and bending moments at various locations is found out. The max shear stress and bending stress are computed. The deflection of CST under these conditions is then computed.

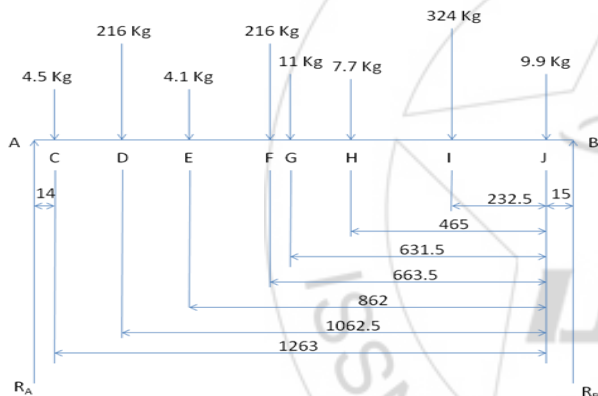
5.6 Calculations

Shear Force:

Considering lateral acceleration of 3g, loads at different rows are:

$$\begin{aligned} F_C &= 4.5 \times 3g = 132.3 \text{ N} \\ F_D &= 216 \times 3g = 6350.4 \text{ N} \\ F_E &= 4.1 \times 3g = 120.54 \text{ N} \\ F_F &= 216 \times 3g = 6350.4 \text{ N} \\ F_G &= 11 \times 3g = 323.4 \text{ N} \\ F_H &= 7.7 \times 3g = 226.38 \text{ N} \\ F_I &= 324 \times 3g = 9525.6 \text{ N} \\ F_J &= 9.9 \times 3g = 291.06 \text{ N} \end{aligned}$$

$$R_A + R_B = 23320.08 \text{ N}$$



Taking moments about B,

$$R_A \times 1292 = (4.5 \times 1278 + 216 \times 1077.5 + 4.1 \times 877 + 216 \times 678.5 + 11 \times 646.5 + 7.7 \times 480 + 324 \times 247.5 + 9.9 \times 15) \times 9.81$$

$$R_A = 10,918.775 \text{ N}$$

$$R_B = 12,401.305 \text{ N}$$

5.6.1 Shear Force at Different Sections:

- Between B & J = 12,401.305 N
- Between J & I = 12,401.305 - 291.06 = 12,110.24 N
- Between I & H = 12,110.24 - 9525.6 = 2584.64 N
- Between H & G = 2584.64 - 226.38 = 2358.26 N
- Between G & F = 2358.26 - 323.4 = 2034.86 N
- Between F & E = 2034.86 - 6350.4 = -4315.54 N
- Between E & D = -4315.54 - 120.54 = -4436.08 N
- Between D & C = -4436.08 - 6350.4 = -10,786.48 N
- Between C & A = -10,786.48 - 132.3 = -10,918.78 N
- From the above it is found that maximum shear force acting on CST is 12,401.305 N. the dimensions of the CST at which the shear force acts is OD=178mm and ID=164mm.

$$\text{Maximum Shear Stress} = \frac{12401.305}{\frac{\pi}{4} \times (178^2 - 164^2)} = 3.2978 \text{ N/mm}^2$$

5.6.2. Bending Moment

As shear force changes its sign at F, bending moment will be maximum at 'F'.

$$BM \text{ at 'F'} = (12401.305 \times 678.5) - (9.9 \times 663.5 + 324 \times 431 + 7.7 \times 198.5 + 11 \times 32) \times 3g$$

5.6.3. Maximum Bending Stress

Applying bending moment equation for the beam.

$$M/I = \sigma/y$$

$$I = \frac{\pi}{64} \times (178^4 - 164^4) = 13768080.86 \text{ mm}^4$$

$$\sigma_{\max} = M/I \times y_{\max} = 26.247 \text{ N/mm}^2$$

5.6.4. Maximum Bending Stress = 26.247 N/mm²

The principal stresses computed from the bending stress and the shear stresses are

$$\sigma_1 = 26.6595 \text{ N/mm}^2 \text{ and } \sigma_2 = -0.4035 \text{ N/mm}^2$$

Applying the **Von-Mises Henky Failure criterion**

$$S_y = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + \sigma_1^2 + \sigma_2^2}{2}} = 26.863 \text{ N/mm}^2$$

Material of CST is aluminum alloy whose UTS = 430 N/mm²

Since the working stresses calculated above are very less and within the permissible limits the design is safe.

5.6.5. Deflection

Using Macaulay's method of computing deflection for a beam carrying no of point loads.

The bending moment at any section distance 'x' from 'B' is given by:

$$M_x = EI \frac{d^2y}{dx^2} = R_B x - W_1(x-a) - W_2(x-b) - W_3(x-c) - W_4(x-d) - W_5(x-e) - W_6(x-f) - W_7(x-g) - W_8(x-h) \dots (1)$$

Integrating equ 1 we get

$$EI \frac{dy}{dx} = R_B x^2/2 + C_1 - [W_1(x-a)^2/2] - [W_2(x-b)^2/2] - [W_3(x-c)^2/2] - [W_4(x-d)^2/2] - [W_5(x-e)^2/2] - [W_6(x-f)^2/2] - [W_7(x-g)^2/2] - [W_8(x-h)^2/2] \dots (2)$$

This is the slope equation.

Integrating the slope equation, we get

$$EI y = [R_B x^3/6] + C_1 x + C_2 - [W_1(x-a)^3/6] - \dots [W_8(x-f)^3/6] \dots (3)$$

Using the boundary conditions for the simply supported beam, we get

$$C_1 = 1,744.5 \text{ N-m}^2 \text{ and } C_2 = 0$$

For maximum deflection, $\frac{dy}{dx} = 0$

From equ 2, x = 737 mm from the rear end B.

The maximum deflection occurs at 0.737 m from rear end of CST.

The deflection at x = 0.737 m is computed from the deflection equation from (3) i.e.,

$$EI y = [R_B x^3/6] + C_1 x + C_2 - [W_1(x-a)^3/6] - \dots [W_8(x-f)^3/6]$$

$$E = 0.7 \times 10^5 \text{ N/mm}^2 \text{ and}$$

$$\text{Area moment of Inertia, } I = 13.77 \times 10^6 \text{ mm}^4$$

$$y = -0.578 \text{ mm (negative sign indicates downward deflection)}$$

Maximum deflection of 0.58 mm occurs at a distance of 0.737 m from the rear end of CST.

6. Analysis

Creo Elements/Pro has numerous analysis tools available and covers thermal, static, dynamic and fatigue finite element analysis along with other tools all designed to help with the development of the product. These tools include human factors, manufacturing tolerance, mould flow and design optimization. The design optimization can be used at a geometry level to obtain the optimum design dimensions and in conjunction with the finite element analysis.

Table: Various applications of Finite Element Analysis

<i>Area of study</i>	<i>Equilibrium problems</i>	<i>Eagan value problems</i>	<i>Propagation problems</i>
1. Civil engineering structures	Static analysis of the trusses, frames, folded plates, roofs, shear walls.	Natural frequencies and modes of structures.	Propagation of stress waves.
2. Aircraft structures	Static analysis of wings, rockets, fuselages, fins and missile structures.	Natural frequencies flutter and stability of aircraft, rocket.	Response of aircraft structures to random loads, dynamic response of aircraft.
3. Hydraulic and water resources engineering	Analysis of potential flows, free surface flows, viscous flows.	Natural periods and modes of shallow basins, lakes and harbors.	Analysis of unsteady state fluid flow and wave propagation problems.
4. Nuclear Engineering	Analysis of nuclear pressure vessels and containment structures.	Natural frequencies and stability of containment structures.	Response of reactor containment structures to dynamic loads.
5. Mechanical design	Stress concentration problems, stress analysis of pressure vessels, pistons, gears.	Natural frequencies and stability of linkages, gears and machine tools.	Crack and fracture problems under dynamic loads.

7. Finite Element Method

7.1. Introduction to FEM

The finite element method represents an extension of the matrix methods for the analysis of framed structures to the analysis of the continuum structures. The basic philosophy of the method is to replace the structure of the continuum having an unlimited or infinite number of unknowns at certain chosen discrete points. The method is extremely powerful as it helps to accurately analyze structures with complex geometrical properties and loading conditions. In the infinite method, a structure or continuum is discretized and idealized by using a mathematical model which is an assembly of subdivisions or discrete elements, known as finite element, are assumed to be interconnected only at the joints called nodes. Simple functions such as polynomials are chosen in terms of unknown displacements at the nodes to approximate the variation of the actual displacements over

each finite element. The external loading is also transformed in to equivalent forces applied at the nodes. Next, the behavior of each element independently and later as an assembly of these elements is obtained by relating their response to that of the nodes in such a way that the following basic conditions are satisfied at each node:

- Equations of equilibrium
- The compatibility of displacements
- The material constitutive relationship

The equations, which are obtained using the above conditions, are in the form of force-displacement relationship. Finally, the force-displacement equations are solved to obtain displacements at the nodes, which are the basic unknowns in the finite element method.

The basic idea in the finite element method is to find solution of a complicated problem by replacing it by simpler one. Since a simpler one in finding the solution replaced the actual problem, we will be able to find only an approximate solution rather than exact solution. In finite element method, it will often be possible to improve or refine the approximate solution by sending more computational effort.

This is a numerical solution for obtaining solutions to many of the problems encountered in engineering analysis. In this method, the body or the structure may be divided into small elements of finite dimensions called finite elements. The original body or continuum is then considered as assemblage of these elements connected at a finite number of joints called nodes.

7.2. Engineering Applications of FEM:

The general nature of its theory makes it applicable to a wide variety of boundary value problems in engineering. A boundary value problem is one of which a solution is sought in domain (region) of a body subject to the satisfaction of the prescribed boundary (edge) condition of the dependent variable of their derivatives. Mostly all engineering problems which are illustrated in the table 4.1 of the finite element method comes under three categories of boundary problems, namely

1. Equilibrium or steady state or time independent problem,
2. Eagan value problem,
3. Transient or propagation programs

7.3. Finite Element Analysis

Finite element analysis was first developed for the use of aerospace and nuclear industries where the safety of structure is critical. Today growth in the usage of method is directly attributable to the rapid advances in computer technology. As a result commercial finite element packages exist that are capable of solving the most sophisticated problems, not just in structural analysis, but for wide range of phenomena such as steady state and dynamic temperature distributions, fluid flow and manufacturing processes such as injection molding and metal forming.

Finite element analysis is used in new product design, and existing product refinement. Modifying an existing product or structure is utilized to qualify the product or structure for

a new service condition. In case of structural failure, Finite Element Analysis may be used to help in determining the design modifications to meet the new conditions.

7.4. Features of ANSYS Software

The following are the special features of ANSYS software:

- It includes bilinear elements.
- Heat flow analysis, fluid flow and element flow analysis can be done.

Graphic package and extensive preprocessing and post processing.

The following shows the brief description of steps followed in each phase:

8. Analysis of Scaffolding Structure Using ANSYS

8.1 Pre-Processor Stage

Importing the model into ansys:

- File ☐ Import ☐ IGES (selecting the scaffolding 'structure. iges' file that has been exported from PRO/E)

Type of analysis

- Preferences ☐ Structural
- Pre-processing of the model:
- Preprocessor ☐ Element type ☐ Add (select 'solid 10node187')
- Preprocessor ☐ Material Properties ☐ Material Models ☐ Structural ☐ Linear ☐ Elastic ☐ Isotropic ☐ EX=70000 and PRXY=0.33
- Preprocessor ☐ Modeling ☐ Operate ☐ Booleans ☐ Overlap ☐ Volumes ☐ Pick all
- Dividing the surface area of the holes into equal quarters:
- Preprocessor ☐ Modeling ☐ Create ☐ Key points ☐ On Line w/Ratio

Now selecting the semicircular line attached to one surface area of the circular plate and the ratio of '0.5' is given. Thus a new key point is created exactly at center of the circumference of the hole.

- Preprocessor ☐ modeling ☐ Create ☐ Lines ☐ Straight lines

Currently picking the above created key points a line is formed.

- Preprocessor ☐ modeling ☐ Operate ☐ Booleans ☐ Divide ☐ Area by Line

Now the inner half surface area of the hole is divided into two equal areas. This process of area division is carried out all over the model.

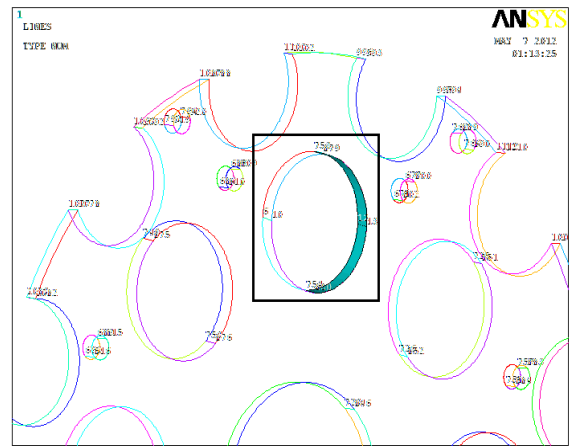


Figure 6: Division of surface area.

Thus the surface areas are divided into equal quarters. This procedure is repeated for all the circular holes present in the model

8.2 Meshing of the Model

- Preprocessor → Meshing → Mesh Tools
- Global Set → Solid 10node187 → Ok
- Check Box → Smart size 'on' to size 6
- Size Controls → Global Set → Element Edge Length= 5
- Mesh → Volumes → tet, free → Mesh → Pick all.

Now the meshing of the complete model will be done. In the model that has been imported has finally produced about 1106223elements and 1823667

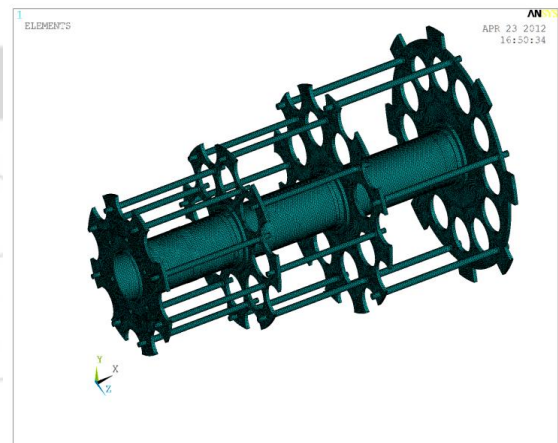


Figure 7: Element plot of the model. Nodes

8.3 Coupling

- Preprocessor → Coupling/ Ceqn → couple DOFs
- Picking the nodes at the starting and ending plates and coupling to the respective nodes on Central Support Tube as these solid plates that should be attached to CST. Coupling is done due to the minor errors in the assembly of the final model so as to avoid incorrect results.

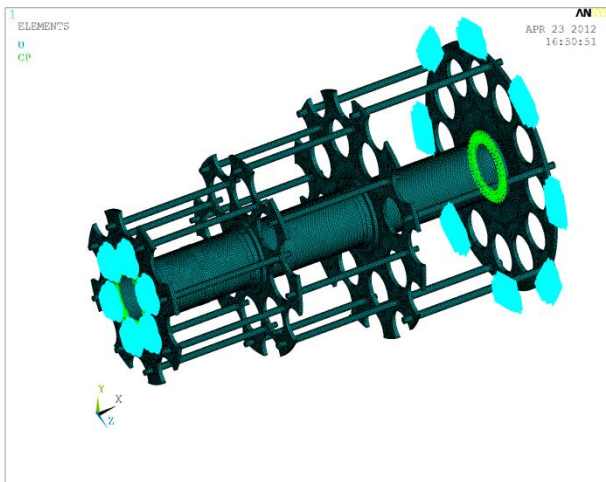


Figure 8: Couplings and Constrains

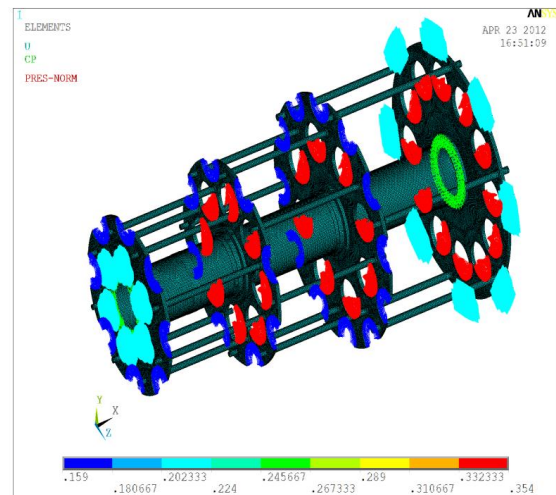


Figure 9: Element Load Plot

Solution Stage

8.4 Application of Loads and constrains

- Solution → Define Loads → Apply → Structural → Displacement → On Nodes

Selecting the nodes present near the circular hole on the starting plate using a circle and the rectangular extrusion on the final or ending plate using a polygon. The displacement of these nodes is arrested in all Degrees Of Freedom (All DOF).

- Solution → Define Loads → Structural → Apply → Pressure → On Areas

Now calculating the pressure that is applied on the surface areas of the holes at an acceleration of '3G' using the below equations.

$$force = \frac{1}{2} \times wt \text{ of each munition} \times 3 \times g$$

$$pressure = \frac{force}{area}$$

Where,

Wt of each munitions = 27kgs.

Acceleration due to gravity, $g = 9.81m/s^2$

Area can be measured directly from the assembly model.

$$force = \frac{1}{2} \times 27 \times 3 \times 9.81 = 397.3$$

Pressure on 1st plate:

Area = 2269.77mm²

Pressure = 0.175N/ mm²

Pressure on 2nd plate:

Area = $\frac{2248}{2} = 1123.05mm^2$ for each quarter.

= 2374.32 mm² for each half section on the circumference.

Pressure = 0.3538N/ mm² for each quarter.

= 0.167N/ mm² for half sections on the circumference.

Pressure on 3rd plate:

Area = 1125.98 mm² for each quarter.

= 2502.25 mm² for each half section on the circumference.

Pressure = 0.3528N/ mm²

= 0.1588N/ mm²

Pressure on 4th plate:

Area = 1129.28 mm²

Pressure = 0.3518N/ mm²

9. Results

Deflection of Scaffolding Structure:

- General postproc → Plot Results → Deformed Shape.
- Max Deflection (DMX) = 0.225mm

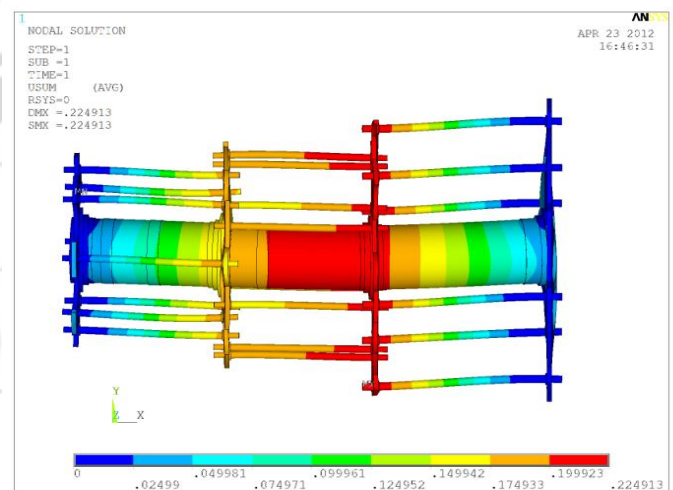


Figure 10: Deflection Plot.

Von-Mises Stress Failure Criteria

- General Postproc → Plot Results → Countr Plot → Nodal Sol'n → stress → von mises stress solution.
 - Minimum Stress (SMN) = 0.179e-05
 - Maximum Stress (SMX) = 193.198

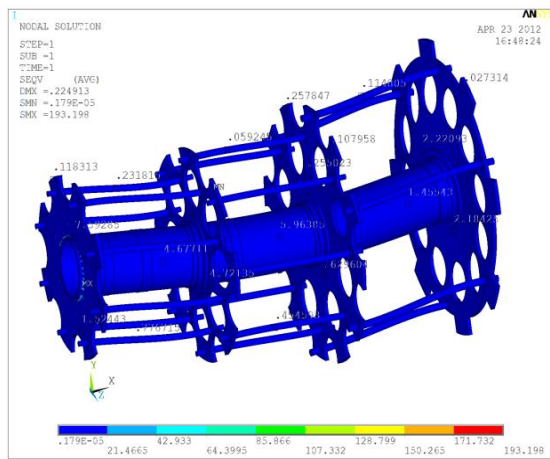


Figure 11: Deflection Plot.

10. Conclusion

Modeling of scaffolding structure is carried out using Pro/E and analysis by using ANSYS. Analysis is done for the pressure loads due to weight of munitions on the scaffolding structure. The project infers that the maximum deflection for the scaffolding structure is less than 0.7 mm with factor of safety of 2. The analysis using ANSYS has been done to the most preferred solution which is, the maximum deflection is 0.225 mm. This value approximately equal to the solution that is obtained in theoretical method and so the result is favorable. Hence the 12-8-8 (Total 28) arrangement of munitions can also be put in practice.

References

- [1] Roark, R. a. (1975). *Formulas for Stress and Strain* (5th edition). New York McGraw-Hill.
- [2] Flight Dynamics Principles, Second Edition: A Linear Systems Approach to Aircraft Stability and Control by M. V. Cook Publisher: Butterworth - Heinemann; 2nd edition (September 28, 2007)
- [3] Literature by Research Center Imarat, DRDO.
- [4] Timoshenko, S. (1930). *Strength of materials, Part I*, D. Van Nostrand Company.
- [5] <http://brahmos.com>
- [6] *Strength of Materials by Ramamrutham.*
- [7] <http://www.aerospacematerials.com.au>
- [8] <http://www.ndt.net>
- [9] <http://www.wikipedia.com>
- [10] <http://engg.geometricglobal.com>