NAFED08 :: Design Competition

Guidelines for the design competition

- Participants can choose any **one** of the following problems.
- Participants are required to perform finite element modelling and analysis exclusively using FEAST.
- Participants are expected to follow best practices in finite element analysis.
- Proper reasoning should be given for assumptions.
- The main criteria of design evaluation will be based on least mass of the hardware and end results.
- Mechanical properties of the material chosen should be taken from standard sources and proper references has to be cited.
- Consistent units need to be used for the analysis.

Design Problem No. 1

In future, ISRO may venture into Mars lander missions involving a re-entry module with outer surface dimensions shown in Figure 1. The module is assumed to be constructed using AA2219 T87 aluminium alloy material, with outer layers comprising suitable ablatives and thermal protection systems. Assume re-entry module is hermetically sealed and temperature inside the module is maintained at 25^0C during launch. Total mass of the packages inside the re-entry module is approximately 425 kg which is uniformly distributed throughout the module. The re-entry process incorporates air braking and propulsion-assisted deceleration systems to manage descent. During re-entry, the module will experience significant heat flux at the outer surface due to aerodynamic heating as shown in Figure 2 and Table 1, along with substantial mechanical loads arising from inertia and pressure forces.

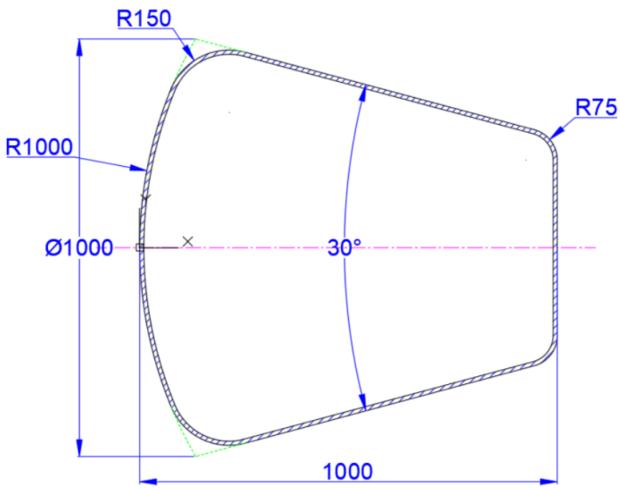

Figure 1: Dimensions of the re-entry module

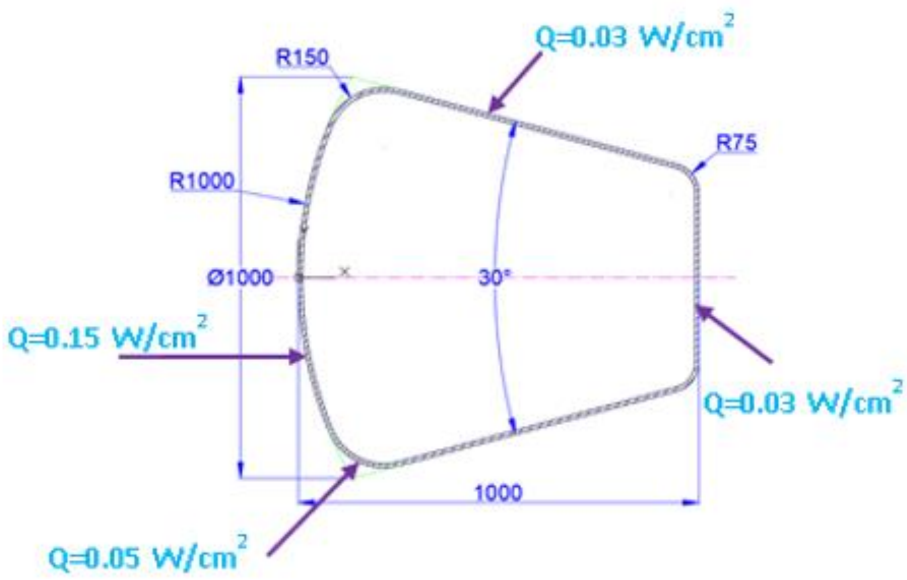

Figure 2 Heat flux distribution in the module

Table 1 Heat flux distribution along the module

Location	Heat Flux, W/cm^2
R1000 (Front region)	0.15
R150 (Shoulder)	0.05
Cone	0.03
R75	0.03
Aft region	0.03

The objective is to design the metallic portion of the re-entry module with a construction (not necessarily a monocoque construction) with minimum mass considering the combined effects of aerodynamic heating and mechanical loading (impact loading on Mars surface need not be considered).

For the analysis, participants can assume that the ablative and thermal protection system layers are intact, so the focus is solely on the metallic layer's performance. A convective heat transfer coefficient of 5 W/m^2K is applicable on both inner and outer surfaces of the module. Mars' atmospheric conditions are assumed to be 6 mbar pressure and a temperature of -60°C. For thermo-structural analysis, fixed boundary condition can be assumed at the periphery of the smaller diameter.

Mass of the packages may be simulated for analysis by applying **non-structural mass properties** at the inner surface of the module. Maximum inertial load experienced by the module is 20g at 30^0 with respect to the x axis.

The analysis process involves the following steps:

- Perform a structural analysis only with mechanical loads to finalize the initial construction of the module.
- Conduct a heat transfer analysis to determine the temperature distribution across the module.
- Map the resulting temperature distribution onto the structural model along with mechanical loads to perform a thermo-structural analysis.
- Estimate the margin of safety based on the results of thermo-structural analysis against yield stress of the material.

Hint: Use Shell element for analysis

Design Problem No. 2

Launch vehicles are subjected to both static and dynamic loads during operation. Dynamic loads arise from factors such as engine ignition, thrust transients, stage separations etc. The structural components and subsystems of a launch vehicle must endure these loads without compromising structural integrity or operational functionality. To ensure this, all subsystems undergo qualification through vibration tests.

Axial vibration tests are typically conducted using an electrodynamic shaker with a 1-meter diameter armature. However, the hardware to be tested often has a larger diameter interface, such as 3 meters. To bridge this size difference, a vibration fixture must be designed to interface the test article with the shaker. Schematic of the fixture is shown in Figure *3*. **The goal is to design a lightweight metallic fixture** that maintains high stiffness, ensuring it meets the following requirements:

- The first axial or torsional vibration mode of the fixture along with the test article must have a frequency greater than 110 Hz, whereas the first lateral vibration mode should be greater than 20 Hz.
- Manufacturability of the fixture through casting should be ensured.
- The height of the fixture can be different from 1m (refer Figure *3*).

For this design, the test article can be assumed to be a **rigid** hollow cylinder with the following specifications:

- Outer Diameter: 2.8 m
- Height: 2.2 m
- Outer Flange Diameter:3.2 m
- Mass: 1.85 tonnes
- Center of gravity: 1.1 m from the cylinder base
- Interface with the fixture: 90 M12 bolts (equi-spaced) at pitch circle diameter of 3m

The shaker interface includes 8 M10 bolts (equi-spaced) each on the pitch circle diameters of 711 mm, 559 mm, 406 mm and 305 mm.

The design process should involve performing a free vibration analysis to determine the fixture's natural frequencies and mode shapes along with the test article. The bolt locations at the interface between the fixture and the shaker armature can be assumed as fixed, while the interface between the fixture and the test article can be modelled using node merging or tying constraints.

Let us assume that the designed fixture is also used for lateral tests with the input as a sine sweep of 1g acceleration from 5Hz to 100Hz at the base of the fixture. Find the responses at the extreme node at the top of the test article along the direction of excitation. 2% of critical damping can be assumed for all the modes.

The analysis process involves the following steps:

- Perform a free vibration analysis to design a fixture for axial and lateral vibration test.
- Perform a frequency response base excitation to find the lateral responses.

Hint:

1. No need to model the shaker and bolts.
2. To ensure the test article behaves as a rigid body during the analysis, its material properties can be adjusted to increase stiffness. Specifically, increasing the Young's modulus of the test article material by a factor of 1000 will significantly reduce deformation making it effectively rigid in the simulation.
   For example:
   - Original Young's Modulus (E): $E_{original}$
   - Modified Young's Modulus (E'):
     $E' = 1000 \times E_{original}$

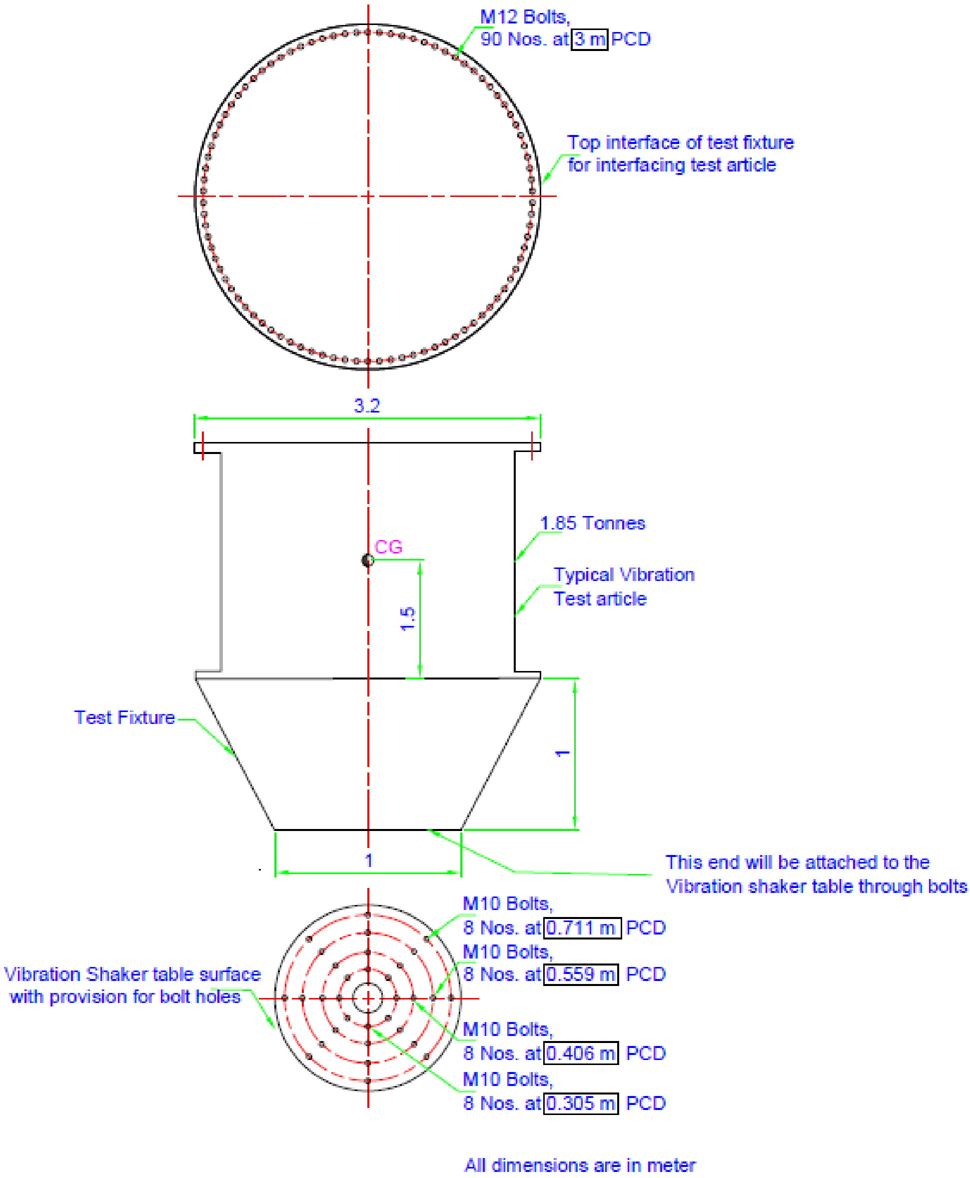

Figure 3: Schematic of Vibration Fixture