1. INTRODUCTION

The Integrated Mechanical System, ported on a mobile carrier Vehicle, is designed to transport, articulate and launch a 10-m long Canisterized Object. The Mechanical System mainly consists of a Tilt Beam that holds a Canister through its various intermittent supports. The Canister houses a 10-m long Object. The Tilt Beam is connected to a Base Frame by means of a Hinge connection at its rear end, Transportation Supports at front end and a pair of Tilt Cylinders in between. The Base Frame is adequately bolted to the chassis of the carrier Vehicle, thus rendering mobility to the Mechanical System. Besides, there are various other accessories like the Outriggers, Support Table etc., which assist in various modes of operation of the System, viz., transportation, articulation and launch.

During transportation, the Tilt Beam is locked to Base Frame by means of Transportation Supports and rear Hinge, while the Tilt Cylinders are inactive. The Base Frame in turn is bolted to Vehicle chassis. At the operation site, the front and rear Outriggers, provided on the Base Frame, are lowered in position so that the entire Vehicle weight gets suspended on the Base Frame. The Transportation Supports are disengaged and the Tilt Cylinders take the load. The Tilt Beam is articulated from horizontal to vertical position with the help of Tilt Cylinders. In the final launch position, the Support Table, which forms the rear part of the Tilt Beam, is lowered and grouted to the ground. A time varying thrust is provided to the undersurface of the Object. Once the exerted thrust exceeds a threshold value, the Object lifts up and finally emerges out of the Canister.

Preliminary design of the Integrated Mechanical System was done using the conventional design methods. These methods involve component level design and often disregard the exact stiffness distribution of the System. Besides, the distribution of forces between various interacting members is sometimes not known precisely. Conventional design is, therefore, quite approximate. Moreover, the exact dynamic response of a complicated system such as the present system is grossly unknown till the actual launch operation.

It was, therefore, imperative to numerically simulate and establish the structural adequacy and stability of the Integrated Mechanical System under various modes of its operation, viz., transportation, articulation and launch and thus confirm the design before attempting an expensive and irreversible actual launch operation.

2. THEORY

Numerical simulation is based on the finite element analysis, which was done to obtain the distribution of stresses and deflections in the Integrated Mechanical System and subsequently assess its structural adequacy and stability in transportation, articulation and launch modes. Important aspects of analysis are summarized below.

The Integrated Mechanical System is a complicated assembly of numerous components with inherently involved boundary conditions. Dynamic analysis warrants a simplified model yet capturing the physics of the system. Thus an equivalent model of the Integrated System was generated such that its mass and stiffness distribution corresponds to that of the actual system. Following elements were used.
a) BEAM44 elements were used to model Tilt Beam and its supports, Base Frame, Outriggers, Transportation supports and Tilt cylinders.

b) Weight of the carrier Vehicle was represented by MASS21 elements and its distribution ensured through rigid BEAM44 elements adequately connected to the Base Frame at points of U-bolts.

c) Canister was modeled using SHELL63 elements.

d) Object was modeled using rigid BEAM44 elements and MASS21 elements.

In the typical growth schedule, a thermal annealing of the Fe-catalyst layer was carried out in the presence of a reducing gas - Ammonia, to break it into small islands, and the multiwalled CNT array growth was carried out over this catalyst by catalytic decomposition of acetylene at 750°C. Ammonia was used as the reducing gas during the CNT growth, while Argon gas was used as purging inert gas during the heating and cooling ramps of the growth cycle. The grown CNT arrays were analysed under an SEM.

Canister houses the Object. There is frictional interface between them. In case of static analysis (transportation and articulation modes) when there is insignificant slip at the interface because of high friction, a fairly good approximation can be achieved by defining constrained relationship between the members at the interface. In launch mode (simulated by dynamic analysis), the Object moves relative to Canister, thus contact elements (CONTAC49) were defined at the interface between the Object and the Canister. This ensures adequate transfer of forces across the contact interface as the Object moves inside the Canister and finally leaves it. Asymmetric contact was used with Canister as the target surface. Contact elements require contact stiffness at the interface. This was derived from the relative stiffness of areas in contact, established in a separate analysis.

The Integrated System contains many accessories, which are not needed for finite element analysis. Neglecting these details in the finite element modeling however, causes change in the self-weight of the system. This was made up by changing the material density such that the weight of each component of the model is equivalent to its actual weight in the Integrated System. Thus the mass and its distribution of the finite element model are equivalent to that of the Integrated System.

Boundary Conditions are different in different modes of operation and are summarized below.

(a) Transportation Mode: In this mode, all attachments over the Vehicle need to be analyzed against the specified loading. Vehicle in any case is designed to carry the Integrated System. Thus the rigid Beam elements and Mass elements representing the Vehicle chassis were removed from the model. All U-bolts locations, from where the load of the Integrated System is transferred to the vehicle have been constrained against linear translation in the three coordinate directions.

(b) Articulation Mode: In this mode, the Object was modeled using rigid BEAM44 elements and MASS21 elements. Loading Conditions are as follows:

(a) Transportation Mode: Self-weight of the Integrated System was increased to account for vehicle deceleration, ground undulations and centrifugal force on a curve. Following values of ‘g’ were specified.

\[
g_x = 0.5 \times g = 0.5 \times 9800 = 4900 \text{ mm/s}^2 \text{ (across the chassis)}
\]

\[
g_y = 2.0 \times g = 2.0 \times 9800 = 19600 \text{ mm/s}^2 \text{ (vertical direction)}
\]

\[
g_z = 0.5 \times g = 0.5 \times 9800 = 4900 \text{ mm/s}^2 \text{ (along the chassis)}
\]

Corresponding inertia force acts in the opposite direction.

(b) Articulation Mode: Weight of the Integrated System is the only force acting in this mode. This is considered by specifying acceleration due to gravity \( g = 9800 \text{ mm/s}^2 \) in the vertical direction. Static analysis was done for various inclination of Tilt Beam with the Base Frame varying from 0° to 90° at an interval of 10°. When the Tilt Beam makes an angle of 90° with the Base Frame, the specified wind force is also applied as a concentrated force (3011 N) on the Canister at the given center of pressure (5350 mm from the Object tip).

(c) Launch Mode: Loads considered in this mode were the self-weight of the Integrated System and the time varying thrust exerted on the Object.

Static analysis was done for transportation and various positions of the articulation modes under the Boundary and Loading conditions explained above. Non-linear transient dynamic analysis was done to simulate the launch mode. In dynamic analysis, loads being functions of time, are applied in time steps. Since the equations solved in dynamic analysis are of second order, two sets of initial conditions are required. These conditions are that time \( t = 0 \), displacements are those caused by the static application of loads (in this case, the self-weight) and velocity is zero. The first time step of analysis has been used to establish these initial conditions. Subsequent load steps specify the applied thrust as a function of time.

Of primary importance in any dynamic analysis is the selection of integration time step. The time step should be small enough to resolve the motion of the structure. Because the dynamic response of a structure may be characterized as a combination of modes, it is necessary to resolve the highest frequency mode, which contributes to the overall dynamic response. With the Newmark integration scheme, it is recommended that time step be small enough so that at least 20 integration points per cycle are included in the highest mode, which will contribute significantly to the overall structural response.
In a separate model analysis of the Integrated System, first 50 modes were identified to be significant. Frequency of 50th mode is 77.4776.

Thus,
Integration time step = 1/(20x77.4776) =0.000645sec
Value of integration time step = 0.0001 was specified for the analysis.

Another important consideration in dynamic analysis is damping. Although, theoretically damping is not important in shock type of loads because the System would have gained its final configuration even before damping affects the energy balance of the System, still some damping needs to be specified for results to converge. Newmark method has very little numerical damping (0.5%), due to which results often show a significant amount of chatter or high frequency excitation. This is particularly true when a force or displacement is applied suddenly. One way to eliminate these higher modes is by specifying some amount of structural damping. Structural (b) damping is frequency dependant and increases linearly with increasing frequency. Structural damping of 0.5% is not uncommon in structures. This can be translated to a b value from the following expression

\[
\beta = \xi/\pi f
\]

where
x = critical damping ratio. This is defined as the ratio of the actual damping to the critical damping for a particular mode of vibration.

f = dominant frequency of response. Typically, the first dominant mode of the structure is used for this calculation. If b is determined at this frequency, then all higher modes will be damped more. This will generally eliminate the chattering discussed previously.

In the present simulation, critical damping ratio of 2 per cent was assumed. The first dominant mode, identified in the modal analysis of Integrated System, is the 3rd mode. Its frequency is 1.9803 Hz.

Thus substituting in above expression, we get

\[
\beta = 3.21476e-03 \text{ s}
\]

On a P-IV Compaq w/s, the non-linear transient dynamic solution took 9 hrs for successful completion.

4. RESULTS AND DISCUSSIONS

The results of finite element analysis are briefly summarized below

Transportation Mode : The Transportation Supports on Base Frame and the bolts used to connect it to the Transportation support on Tilt Beam develop stresses beyond the allowable values. It was, therefore proposed to strengthen the supports by welding additional 8 mm plates to the Transportation supports on Base Frame and use 32 mm dia bolts to connect the two transportation supports. With these changes, the maximum stress experienced by the Integrated System is 152.417 N/mm² as shown in fig. 2. Component-wise, the stresses in Tilt Beam, Transportation Supports, Canister, Base Frame are 82.617 N/mm², 152.417 N/mm², 69.841 N/mm² and 102.073 N/mm² respectively.

Deflection in this mode is insignificant, maximum being only 5.0473 mm experienced by the Tilt Beam.

Articulation Mode : Different components of the Integrated System develop maximum stresses in different positions. The maximum stresses in various components are as follows

<table>
<thead>
<tr>
<th>Component</th>
<th>Maximum Stress (N/mm²)</th>
<th>Inclination</th>
<th>Angle (°)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tilt Beam</td>
<td>119.208</td>
<td>0°</td>
<td>0°</td>
</tr>
<tr>
<td>Transportation</td>
<td>43.037</td>
<td>0°</td>
<td>0°</td>
</tr>
<tr>
<td>Supports</td>
<td>53.205</td>
<td>10°</td>
<td></td>
</tr>
<tr>
<td>Canister</td>
<td>139.755</td>
<td>0°</td>
<td>0°</td>
</tr>
<tr>
<td>Tilt Cylinders</td>
<td>104.619</td>
<td>0°</td>
<td>0°</td>
</tr>
</tbody>
</table>

Tilt Beam tip deflects to a maximum of 25.891 mm. This occurs in the horizontal position of the Tilt beam just before the articulation starts. The deflection of the Beam tip reduces, as the Beam is made vertical. Each Tilt Cylinder develops a maximum compressive force of 204220 N(θ =0°), which changes to tension when θ is between 70° to 80°. The maximum tensile force experienced by Tilt cylinder is 57122 N, which increases to 64527 N with the specified wind considered along the vehicle chassis. Fig. 3 & 4 show stress distribution in two typical position of the articulation mode. These figures do not include shell elements corresponding to Canister for clear visualization of stresses in the BEAM elements.

Launch Mode : Stresses and deflections in the Integrated System, during launch, are much less compared to that in the articulation mode since now the support table legs are also constrained in addition and therefore transfer the load directly to the ground.

Numerical simulation of launch mode reveals the complete dynamics of the Integrated System as the Object emerges out of the 10 m long Canister. The deformation status of the Integrated System at t=0, just before the Launch. These deformations are essentially due to the self-weight of the System. Maximum deflection, however is only 1.316 mm.

As the Object experiences the time varying thrust, it starts lifting up. The deformation status of the Integrated System at various intervals of time beyond time t=0. The value of DMX in the legend of these figures shows the maximum deformation. Since actual deformations are insignificant, the value of DMX is almost equal to the distance traveled by the Object with respect to its position at time t = 0s. The Object leaves the Canister somewhere between time t=0.67188 and 0.68088 s.

Since the Object accelerates and leaves the Canister in a very short time, the Integrated Mechanical System starts vibrating. The vibration response of the tip of Tilt Beam along the trailer chassis. Maximum displacement of Tilt Beam tip along the trailer chassis is 0.3525 mm. The vibration of the Integrated Mechanical System exerts lateral forces on the Object, which tend to deviate it from its vertical trajectory. The deformation of the Object nose as it comes out of the Canister. At time t = 0.68088 s, when the Object is fully out of the Canister, Nose tip deviates by 1.0222 mm (along the trailer chassis) from its vertical trajectory.
The stresses and deflections in various modes of operation, as discussed above, are well within the allowable limits. The Integrated Mechanical System is, therefore, structurally safe and stable. The Integrated System has been experimentally test verified (under the static loads) and is awaiting the actual launch trials.

5. CONCLUSIONS

If the distribution of mass and stiffness is adequately simulated, every physical System can be represented by means of a simplified finite element model. This becomes more of a necessity if a complicated Integrated System is to be analyzed for its dynamic behaviour. The present paper discusses a simple equivalent model of an Integrated Mechanical System and its finite element analysis to assess the structural adequacy and stability of the System in various modes of its operation viz., transportation, articulation and launch.

The finite element analysis clearly establishes the strength requirements in transportation modes, strength reserves available in various positions of the articulation mode and interaction of forces between various components of the Integrated System and their safety during the launch mode. This numerical effort has not only led to improvement in design but also given confidence for the forthcoming field trial of the System.

ACKNOWLEDGEMENTS

The author is indebted to Director R&DE(Engrs) for establishing excellent computing facilities without which it would have been impossible to carry out the numerical simulation. The author is obliged to Mr Sangam Sinha, Project Director, and his team for providing the required input data for numerical simulation and cooperation extended during the review and interpretation of the results.

REFERENCES

11. RD Cook, David B Malkus, Michael E Plesha, Concepts & Applications of Finite Element Analysis.