Proceedings of the 18th International Symposium on the
Packaging and Transportation of Radioactive Materials
PATRAM 2016
September 18-23, 2016, Kobe, Japan

Paper No. 2037

Why use the sub-modelling technique in a Finite Element Analysis of a nuclear transport package?

Author Chris Berry
Affiliation International Nuclear Services Ltd

Abstract

A nuclear transport package has been analysed using Finite Element Analysis (FEA) under regulatory impact conditions. Embarking on impact analyses can involve large assembled FE models comprising many individual components and the more complicated a model becomes the longer it is likely to take to complete the analysis solution. It is known, that generally, in order to achieve a sufficient level of accuracy in the analysis results, of a particular Region of Interest (ROI), typically means a refinement in the mesh density is required. Furthermore, in some instances the level of fidelity in the FE model falls short of the degree of refinement that is needed to capture the small details in the geometry.

Modelling and simulating the impact of a structure involves calculating large deformations over many small time steps dynamically and the use of sophisticated explicit analysis tools. However, as a consequence, simulation of large structures in explicit analyses can be very time consuming and costly - the analysis time step is directly influenced by the smallest element size in the mesh. It is also the size of the smallest element that limits the stable time step.

Using the sub-modelling technique allows a refined analysis of a ROI of a local part of a larger model in much greater detail than that directly obtained from the original full (global) model, but without the additional time and cost implications of refining the entire structure. The process of performing a sub model analysis is described with the use of the explicit FEA tool, LS-Dyna.

Introduction

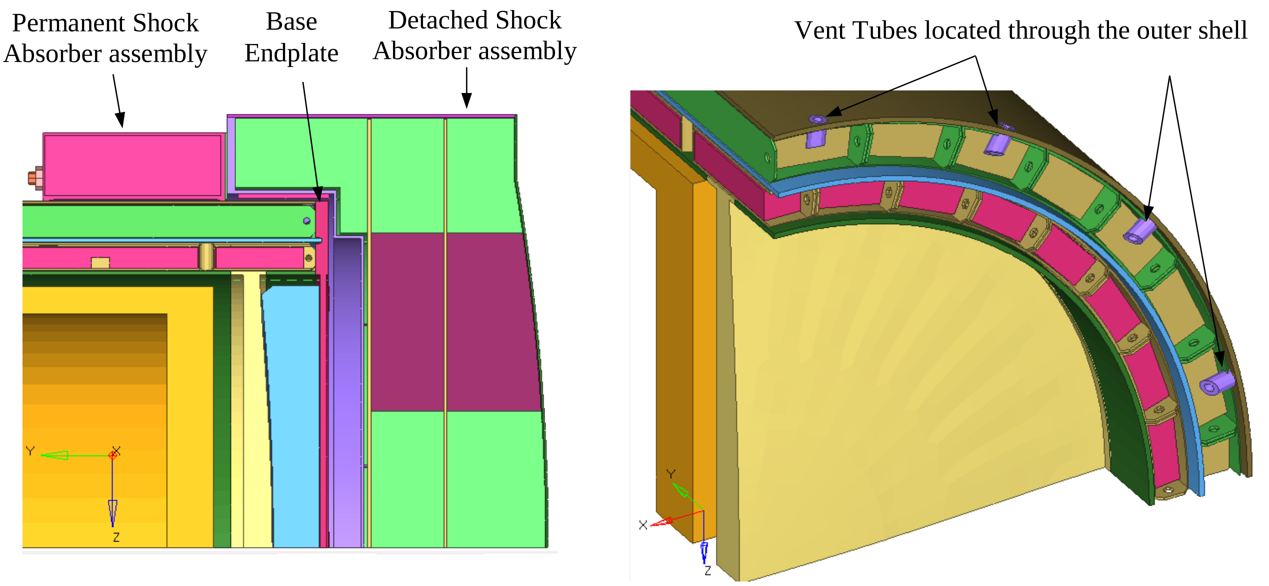
A design proposal was made to include a number of vent tubes in the M4-12 transport package, **Figure 1**, to provide a pressure relief from an internal cavity between the outer and inner shells. The vent tubes allow any potential gas produced from degradation of the neutron shielding material called vitrite, to escape out of the cavity. Such gases could escape through the vent tubes as long as the geometrical integrity of the vent tube itself is maintained. This is considered true under normal operating conditions. However, it is not fully understood how well the vent tube would resist deformation caused by an impact at the base end of the M4-12 package. The design intent is that the vent tube must allow the gas to escape after the worst impact scenario.

Figure 1 – M4-12 RAM Transport Package.

During an accidental fire condition, temperatures may exceed 390^{o}C at which will cause the neutron shielding material to degrade **[1]**. This degradation process of the vitrite causes off-gassing to occur, which may increase the internal pressure in the cavity between the inner and outer shells of the package.

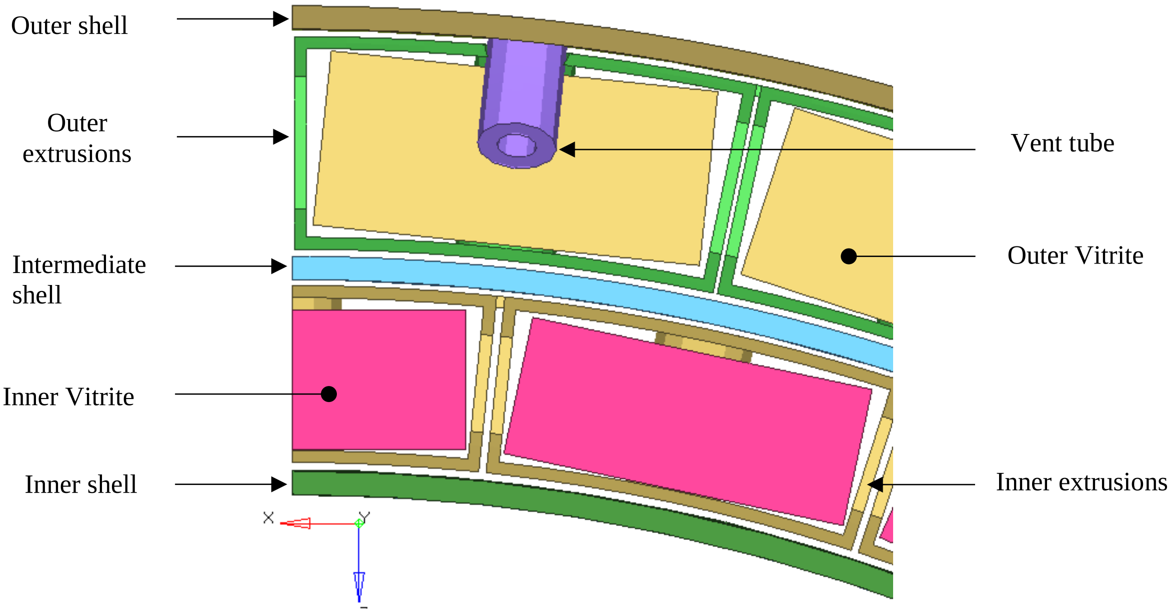
Due to the close proximity of the vent tubes to the base end of the package, a vertical drop of the full package onto its base end shock absorber, is considered to be the worst drop orientation in respect to the amount of damage caused to the vent tube components. There are two reasons for this; firstly, the localised damage would be concentrated at the base end, causing material deformation close to the locality of the vent tubes. Secondly, the adverse effects from inertial forces of the combined aluminum extrusions and vitrite blocks coming into contact with the side wall of the vent tube itself at the time of impact.

Figure 2 shows multiple views of the M4-12 base end assembly with emphasis on the position of the vent tubes. The design proposal was to utilise 15 vent tubes positioned around the circumference of the outer shell, each one located central to every other outer extrusions. As evident in **Figure 2**, the vent tube component is in close proximity to the end of the outer shell edge, close to the base end plate.



(b) **Section view of the M4-12 base end assembly**

(a) **Iso view of the base end internal structure. The Shock absorber and end plate removed for clarity**



(c) **Close-up view of the vent tube and surrounding structure between the inner and outer shells**

Figure 2 – Multiple views of a quadrant section of the M4-12 base end showing the structural assembly and description of the position of the vent tubes.

The purpose of this analysis was to demonstrate that the vent tube would not fail by maintaining its integrity after the M4-12 package is dropped from a regulatory 9m height onto its base end causing impact damage. Failure of the vent tube from impact damage means an inability to provide a pressure relief.

The analysis was carried out by using the sub modelling technique; the creation of a sub model and the process of the sub modelling technique and its benefits are discussed in this paper. A sub model has been used to perform an impact analysis to evaluate the plastic deformation in the vent tube and its immediate surrounding material. The vent tube material was stainless steel grade S316. A further analysis was completed to compare the effects from an alternative material, FV520B.

Modelling Approach

To investigate the integrity of the vent tube and accurately predict its deformation caused by impact damage requires a high level of fidelity in the mesh. A good meshed model of the vent tube and its surrounding material can give the following benefits: firstly, capturing the true geometry and dimensions of the vent tube to drawing. Secondly, to gain a level of confidence in the stress/strain behaviour of the material. This extends to the surrounding material, as it can influence the results of the vent tube component.

Finite Element (FE) models of large structures that contain many components are usually large in terms of model size, consisting of high element counts. This is true for the FE model for the entire M4-12 package that has been used in this analysis. Large FE models that contain many components and consist of a high number of elements usually take a long time to complete the calculation of the analysis. Therefore, it is not always practical to build large FE models of high fidelity. So, careful decisions have to be made in the construction stage of the FE model, which often leads to a compromise - the analyst deciding the optimal performance between analysis run times against the level of accuracy that is required in the results.

Adopting the sub modelling technique makes it possible for a localised region of the FE model to be refined. Increasing the mesh density of a particular detail of the whole structure, improves the definition of the material behaviour in a localised region of interest. In the world of sub-modelling, the larger FE model is referred to as the global model. This is analysed first as a pre-requisite to the creation and running of a subsequent sub model.

As the global model is solved first in a sequential analysis, the sub model analysis relies on specific driving data to be taken from the global model. This driving data is then applied to the sub model. This data is taken from the global model and is imposed on to the sub model as prescribed boundary conditions. In LS-DYNA **[2]** the loading data used were displacements and velocities.

Global Model Description

The FE global model comprises of a reduced quarter symmetry model of the full M4-12 package. **Figure 3** shows a 3D image of the M4-12 package geometry with a quadrant FE model superimposed. This quadrant is the global model.

The global model used in the analysis is based on an existing developed FE model. The lid-end shock absorber and internal furniture have not been included as meshed entities, but they have been considered by distributing their mass across the rigid regions of the model. The base-end shock absorber has been modelled to incorporate the structural components needed to impart transfer of the impact loads into the base end of the package.

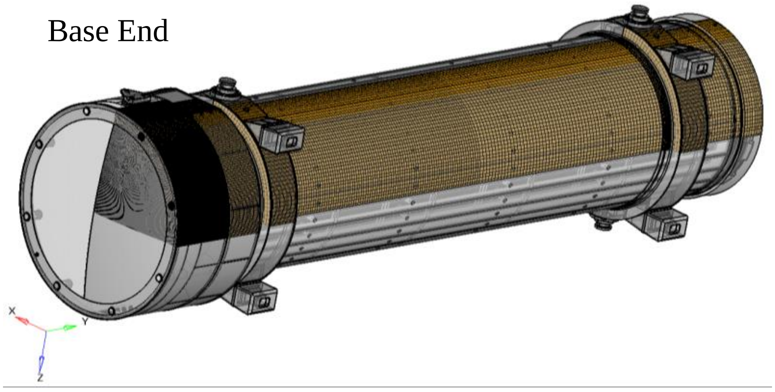
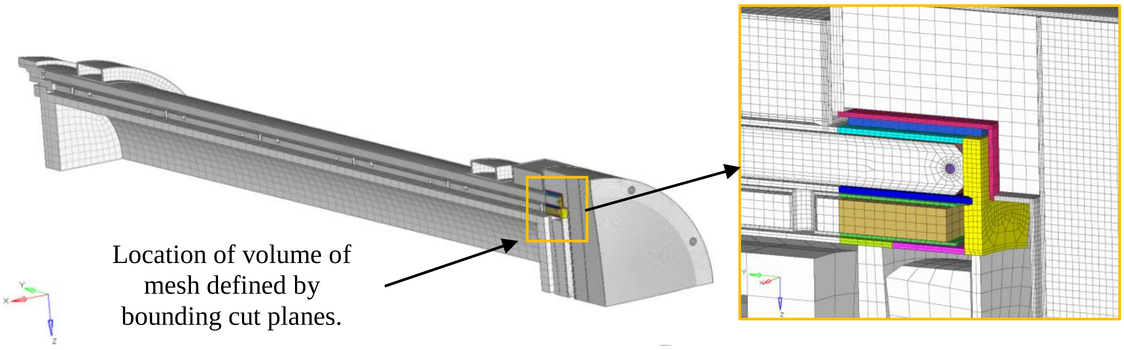
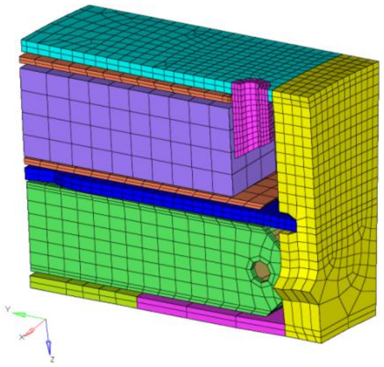


Figure 3 – M4-12 geometry showing a superimposed quadrant global FE model.



(a) M4-12 FE global model

(b) Close-up view of volume mesh



(c) Section view through y-z plane of the global volume mesh

Figure 4 – M4-12 FE Global Model used in the impact analysis.

The FE global model was constructed from 3D brick elements. The only exception was the stiffening plates of the permanent shock absorber, these used shell elements.

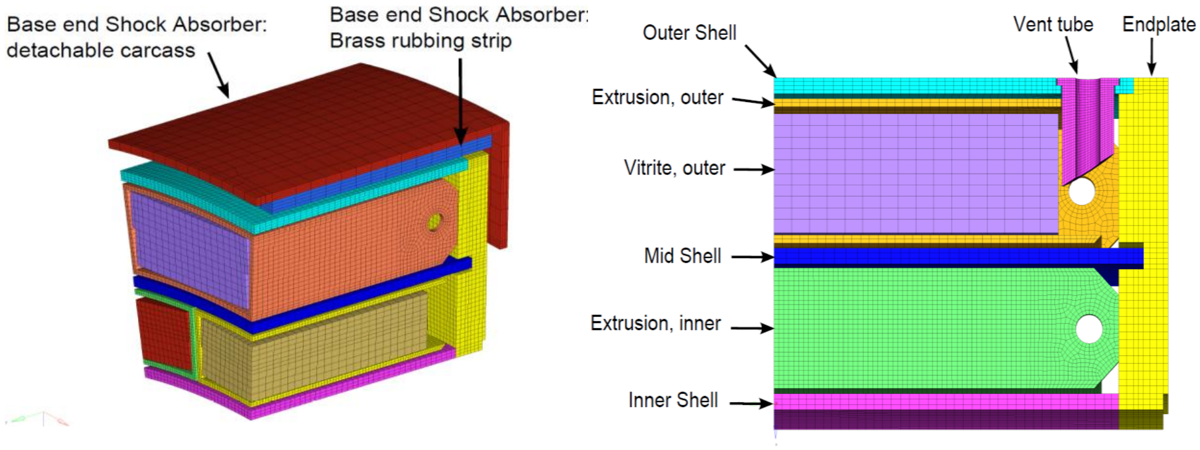
The global model is shown in its entirety in the top left of **Figure 4**. A very small volume of the global model has been utilised to form the creation of the subsequent sub model. The size of the volume has been derived by predefined bounding cut-planes, which subsequently dictates the size of the sub model. The cut planes and thus the size of the volume mesh, have been carefully chosen to include sufficient material at given distances outwards from the vent tube component. It is important to include some of the surrounding mesh, as this will be used in the sub model analysis and refined to allow improved prediction of its material behaviour.

The mesh density of the volume mesh is too coarse and the geometry of the vent tube has been simplified.

Sub Model Description

The FE sub model is shown in **Figure 5**. All sub model components apart from the vent tube, which has been modelled in full, have been cut from the global model. In comparison to the global model, all components of the sub model have much refined mesh densities, especially the vent tube itself. This has been modelled using a high mesh density. The only exception applies to the shock absorber components, which have the same mesh as the global model.

The main dimensions of the vent tube are: a 10mm diameter hole and a wall thickness of 5.6mm. The hole provides an orifice through which the gas, from the potential degradation of the vitrite material, can escape from the internal cavity.



(a) Iso view of the sub model

(b) Section view showing the vent tube, shock absorber omitted for clarity

Figure 5 – FE Sub Model and identification of all its individual components.

Impact Analysis

All analyses were solved using LS-Dyna 7.1.2 **[2]** on a linux cluster in double precision, MPP mode. The global model was solved using 24 CPUs. The sub model was solved using 12 CPUs.

All the pre and post processing was done using Altair HyperWorks version 13.0 **[3]**. Specifically, the FE models and the setting-up of the analysis input files were generated in HyperMesh. All results were processed using HyperView.

All analyses were performed with automatic contact assigned to all component mating faces using the *AUTOMATIC_SINGLE_SURFACE keyword. No friction was assigned in the contact behaviour between mating faces across components, assuming this is the worst case scenario.

In the impact analysis, the M4-12 global model (**Figure 4**), was dropped with the base end facing downwards from a 9m drop height onto a rigid surface, the base end external shock absorber taking most of the impact.

The analysis was simulated for a total duration time of 20 milli-seconds (ms). This is a sufficient termination time as the kinetic energy (KE) reduces to a minimum. Energy balance graphs were recorded for both global and sub models, see **Figure 6**. The KE reaches zero at 14 ms at which point in time the package has come to rest. After 14ms the package gains KE due to the rebound from the rigid surface.

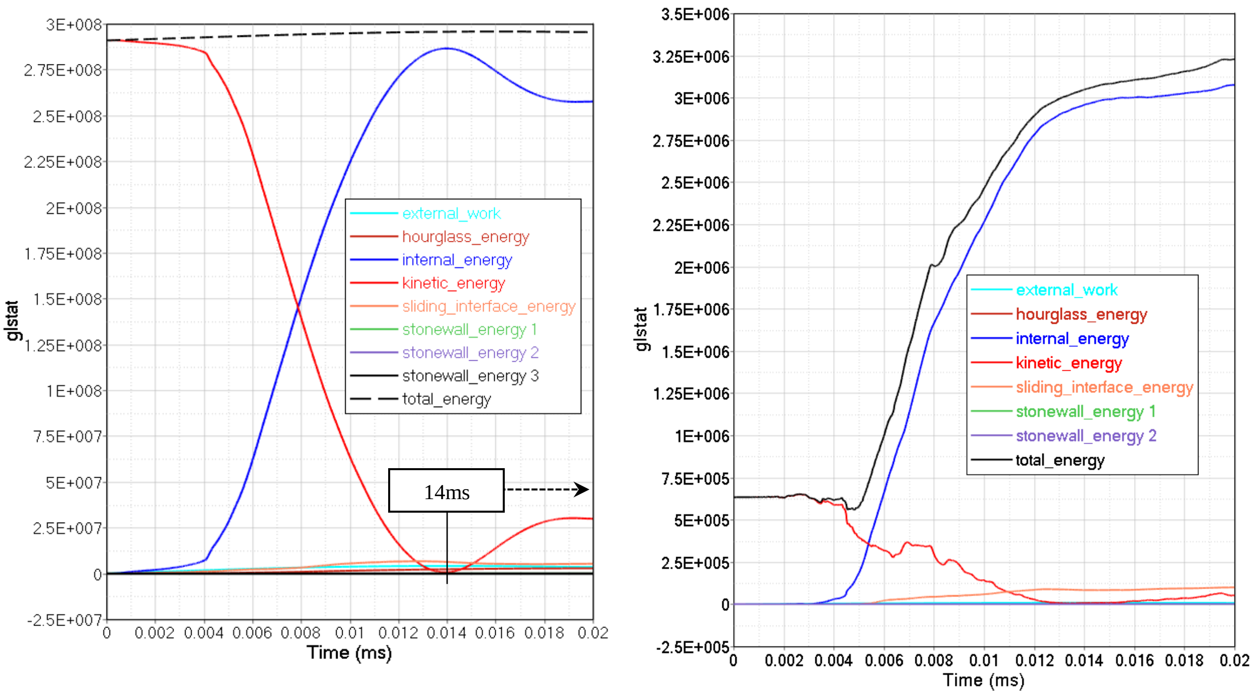


Figure 6 – Energy curves taken from the global model (*left*) and sub model (*right*) for the full analysis time frame: 0 through 20ms.

Materials

All the mechanical properties for all materials have been obtained from the INS Flask Materials Database. The mechanical properties specific to the materials used for the vent tube are listed in **Table 1**. Material type, S316L is the baseline material. Material type, FV520B is an alternative that has been considered for comparison. As the focus is on the vent tube, only the materials associated with this are listed.

Primarily, the yield stress limits between the two material types are of interest. The yield stress value of 780MPa for FV520B is significantly higher compared to 220MPa for S316L. This means that FV520B has the ability to withstand much higher levels of stress, well beyond S316L, before it starts to deform plastically.

In LS-DYNA, all material data assigned to the FE sub model uses the Material Type 24, *MAT_PIECEWISE_LINEAR_PLASTICITY, which is an elasto-plastic material. This material type allows the non-linear behaviour to be surveyed, which is an important requirement in assessing the level of deformation of the vent tube in an impact analysis.

Table 1– Mechanical Properties of Materials used for the vent tube.

Material	**Density** [t/mm3]	**Young's Modulus** [MPa]	**Poisson's Ratio**	**Yield Stress** [MPa]	**Tangent Modulus**
S316L	7.9×10^{-9}	200,000	0.3	220	1,510
FV520B	7.8×10^{-9}	214,000	0.3	780	2,259

Boundary Conditions and Loading

Two types of boundary conditions were applied to the sub model, these can be seen in **Figure 7**. Firstly, a node set consisting nodes lying on the cut plane belonging to solid element faces (depicted by the blue areas) have been constrained in degrees of freedom, Tx, Ry and Rz. The fixed boundary condition was assigned to the node set using *BOUNDARY_SPC_SET. The reason for fixing the nodes in the sub model originates from govenance of the preceding global model - the fixed boundary condition applied in the sub model is consistent with the global model, matching its bounding cut plane from the global quadrant model.

Secondly, two additional node sets were created to apply symmetry boundary conditions to the outer faces of the outer extrusions (highlighted in yellow). These symmetry boundary conditions apply a restraint in the normal direction to the outer faces, taking into account the presence of the opposing material from neighbouring extrusions. The symmetry boundary conditions were modelled by the creation of two flat rigid planar walls via the *RIGIDWALL_GEOMETRIC_FLAT keyword.

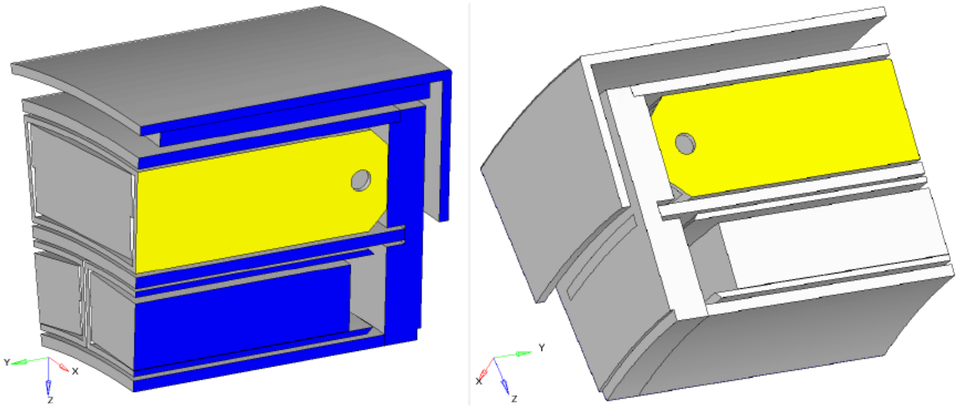


Figure 7 – Identification of symmetry (*yellow*) and fixed (*blue*) boundary conditions applied to the sub model.

The global model analysis was performed with specific loads recorded from the global model analysis and used to drive the sub model. The loading used was in the form of displacements and velocities. These were recorded from the global model across predetermined solid element faces, known as segments. These segments are associated with element faces that lie on chosen cut planes, dictated in the global model. A total of six individual sets of segments were generated for various regions across the cut planes. These segments can be seen in **Figure** 8 for the global model and **Figure 9** for the sub model.

All six sets of segments of the global model were assigned using the keyword: *SET_SEGMENT. The corresponding sets of segments in the sub model analysis were assigned using *INTERFACE_LINKING_SEGMENT.

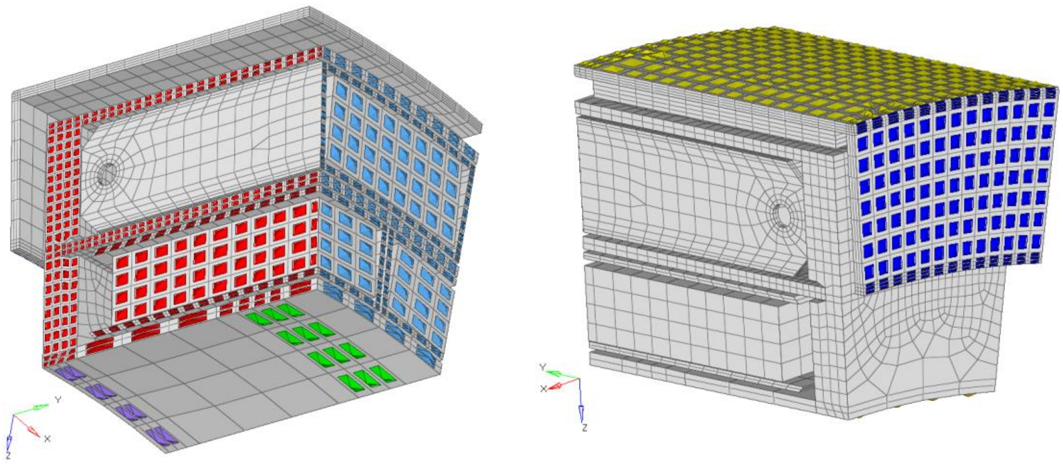


Figure 8 – Extracted volume mesh from the global model, showing the individual, predetermined segments.

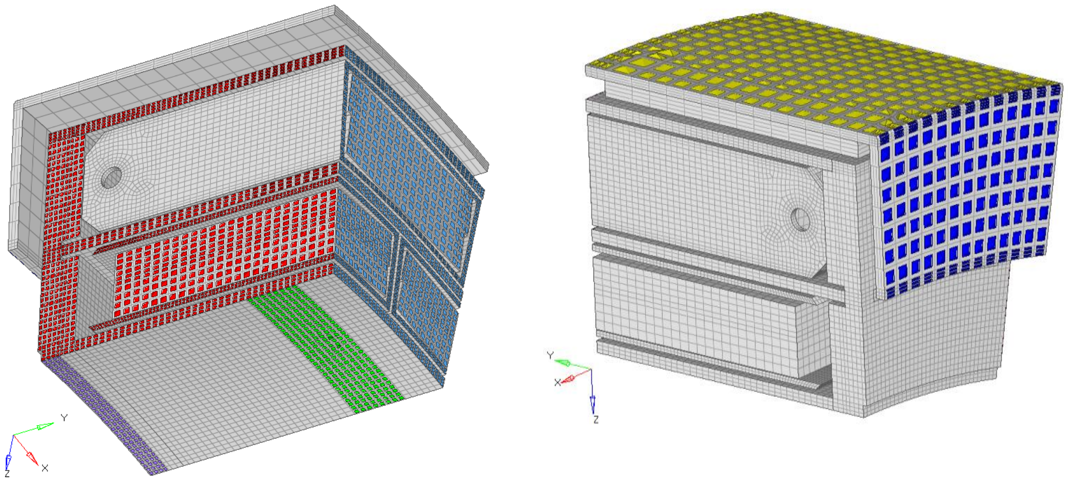


Figure 9 – Sub model: refined mesh showing the individual segments.

The M4-12 package (global model) was dropped vertically from a height of 9m, with the base end facing towards a rigid surface. A velocity of 13,288 mm/s was assigned using the *INITIAL_VELOCITY keyword. The initial velocity was calculated using the following formula: $v = \sqrt{2 * g * h} \quad = \sqrt{(2 * 9810 * 9000)} \quad = 13{,}288\, mm/s$

The model was dropped vertically with the base-end shock absorber impacting onto a rigid surface. The rigid surface was simulated using *RIGIDWALL_GEOMETRIC_FLAT.

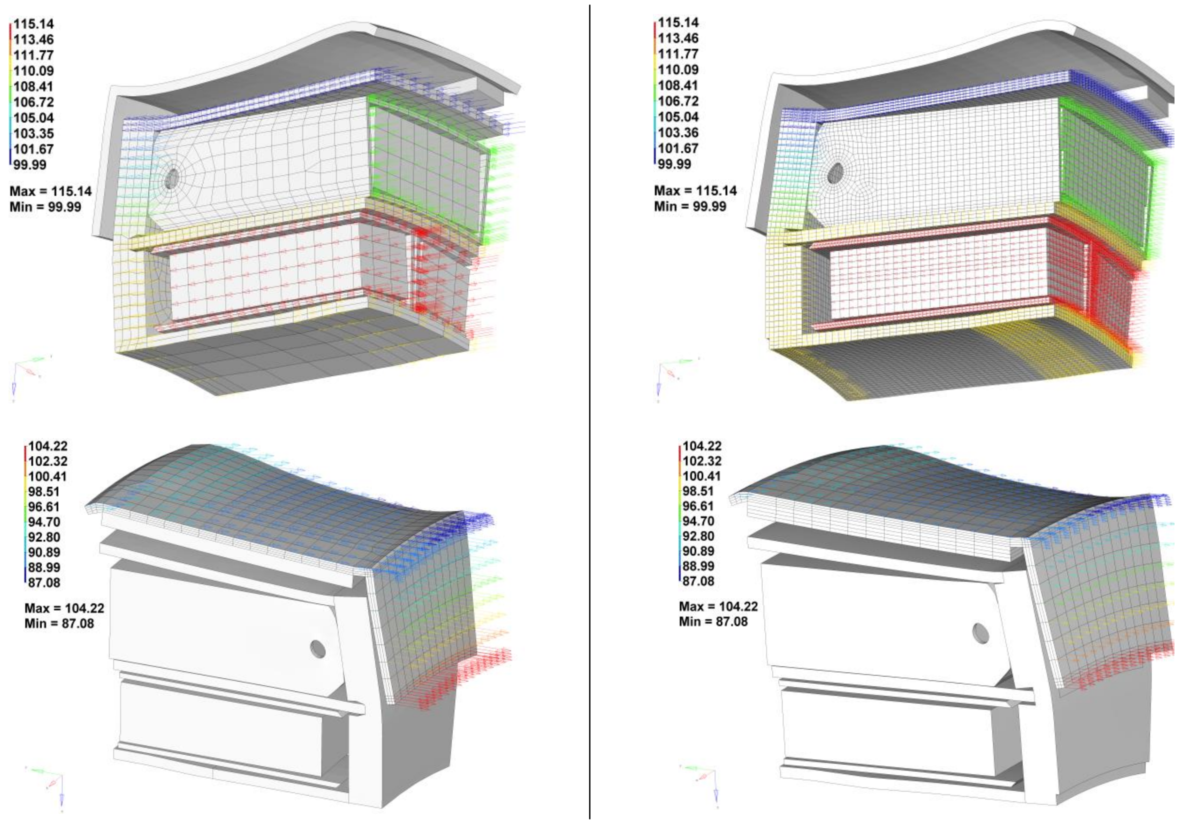
Gravity was assigned to the M4-12 global FE model (**Figure** 4), by the using the *LOAD_BODY_Y keyword in conjunction with a curve to assign gravity force (9810.0m/s^2).

Checking of mapped loads between global model and sub model

It is important to perform a load transfer check to verify that the loads calculated from the global model match with those being applied to the sub model. The loads applied to the sub-model must be the same as on the global model. If there are any differences in the loading between those recorded from the global model and transferred to the sub model, would mean the results will not resemble the loading correctly and render them invalid.

Accurate transfer of loading is vital in order to achieve accurate results in the sub model. A satisfactory method to verify if the loads have transferred between models correctly, is to compare the displacements at all respective cut planes of the sub model with those from the global model. This can be done by displaying the resultant vector values/orientations from both models. This check can be done at the end of the analysis where the last time increment was updated.

Figure 10 provides evidence of this check. The displacements derived from the global model have successfully mapped onto the sub model. The use of vector plots demonstrates that both directions and magnitudes applied to the individual nodes are in agreement and that the load transfer has been performed. A further check should be done at different intervals at different times throughout the analysis to prove the time history.



(a) Global model **(b) Sub model**

Figure 10 – Illustration of the derived displacements from the global model and mapped onto the sub model.

Results

Figure 11 shows contour plots of the effective plastic strain in the vent tube made from material, S316L. This highlights the difference in the plastc strains between the global model and the sub model. The maximum locale of plastic strains in the vent tube component changes location in the refined sub model because of the following differences in the sub model: the true geometry has been captured and the material behaviour is refined due to the improved level in model fidelity.

The distribution of plastic strain in the vent tube is much less with the use of material FV520B, see **Figure 12**.

All plots have been taken at the termination time of the impact analysis of 20 ms. The deformation of the vent tube and the surrounding material can also be seen in these plots.

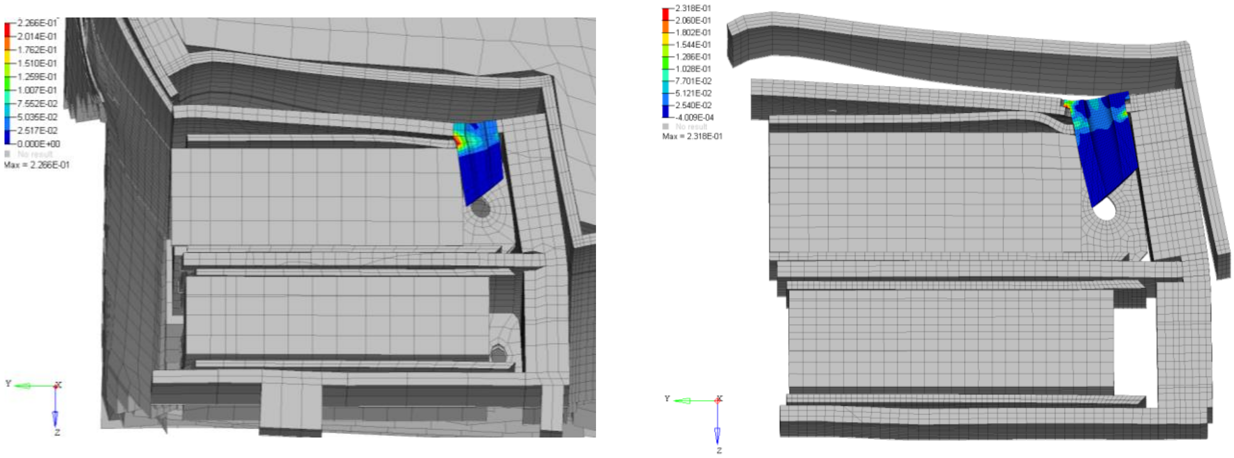


Figure 11 – Comparsion of the effective plastic strain of the vent tube in the global model (*left*) and sub model (*right*).

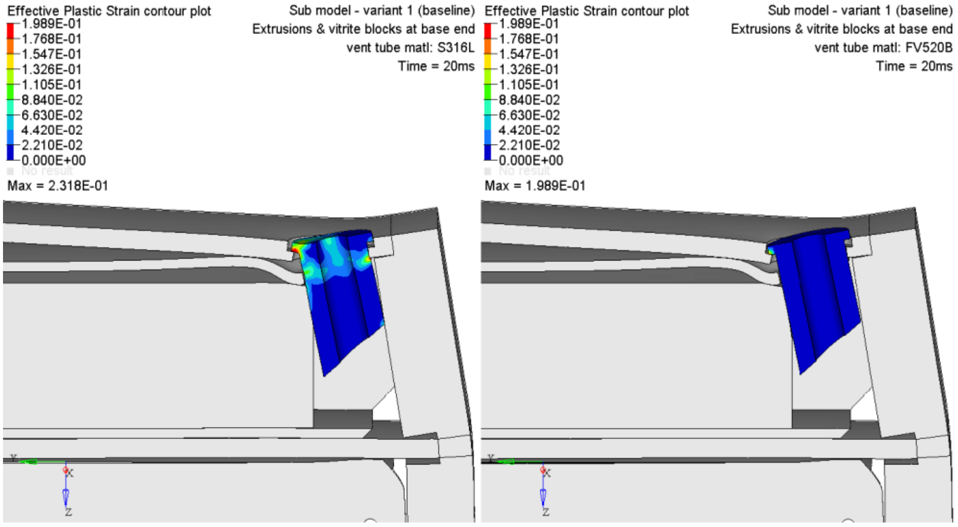


Figure 12 – Sub model results – Comparison of the effective plastic strain in the vent tube using S316L (*left*) versus FV520B (*right*).

Discussion

Material type FV520B for the vent tube offers a much higher resistance to deformation damage caused by impact with low levels of plastic strain across the wall of the vent tube.

The high level of fidelity in the sub model provides a good level of confidence in the results of the vent tube itself and of the immediate surrounding material. Moreover, the sub modelling technique enables the analyst to create a FE model with increased mesh densities, capturing the geometry accurately. It promotes a better close-to-reality structural behaviour of the material.

Other benefits to mention are evident from reduced calculation run times and file sizes produced during analysis. **Table 2** shows the run times for the models presented here. The global model is much larger, in that it contains a higher number of nodes and elements. In comparison, the sub model is 16.5 times smaller than the global model which is not surprising given that the sub model is only a very small region taken from the M4-12 package.

Table 2 - Comparison of model size and file sizes.

Model type	**Input file size** [MB]	**Model count**	**Time to complete analysis**	**D3plot** [GB]	**Min time step**
Global model	233,872	E: 1,310,000 N: 1,779,022	10h 52m	20.3	1.24e-007
Sub model	14,394	E: 79,286 N: 108,531	6h 22m	1.23	9.21e-008

The ratio between the element count and the time taken to complete the analysis are not equal. This comes about because the sub model is made up of a higher fidelity mesh and is purposely created for the reason sub models are used. This is simply to gain an improved response set of results for the region of interest.

The minimum time step is dictated by the critical length of the smallest element in the entire FE model. The smaller the element size, the longer the analysis takes to complete.

Conclusions

The geometrical integrity of the vent tube, after a base end impact of the M4-12 package, is maintained and would not prevent the generation of gas from the degradation of the vitrite from escaping out.

The sub modelling technique was run successfully allowing a much more detailed mesh to be investigated providing high fidelity results in the region of interest.

The material FV520B is the recommended choice for the use of the vent tube.

References

[1] S.F Jagger, 2014, INS3578: IAEA 800oC Thermal Test – Report of the Repeat Test, MH/14/11, Health and Safety Laboratory.

[2] LSTC. LS-Dyna Version 7.1.2.

[3] Altair HyperWorks Version 13.0.