A 9m Drop Test Simulation of a Dual Purpose Cask for Spent Fuel Elements of Nuclear Research Reactors

Miguel Mattar Neto
Carlos Alexandre de J. Miranda
Gerson Fainer
IPEN – CNEN/SP

Rogério Pimenta Mourão
Luiz Leite da Silva
Arivaldo do Sacramento
Hugo Dalle
Cláudio Cunha
CDTN – CNEN/MG

Project Purpose & Partners

Main purpose - to develop methods for the evaluation of modeling and results, in order to apply in future prototypes design, as part of a Latin American multinational project sponsored by AIEA

The project partners are the Institutes IPEN & CDTN (from Brazil) and South American countries with research reactors, to qualify a shipping cask for their irradiated fuel elements

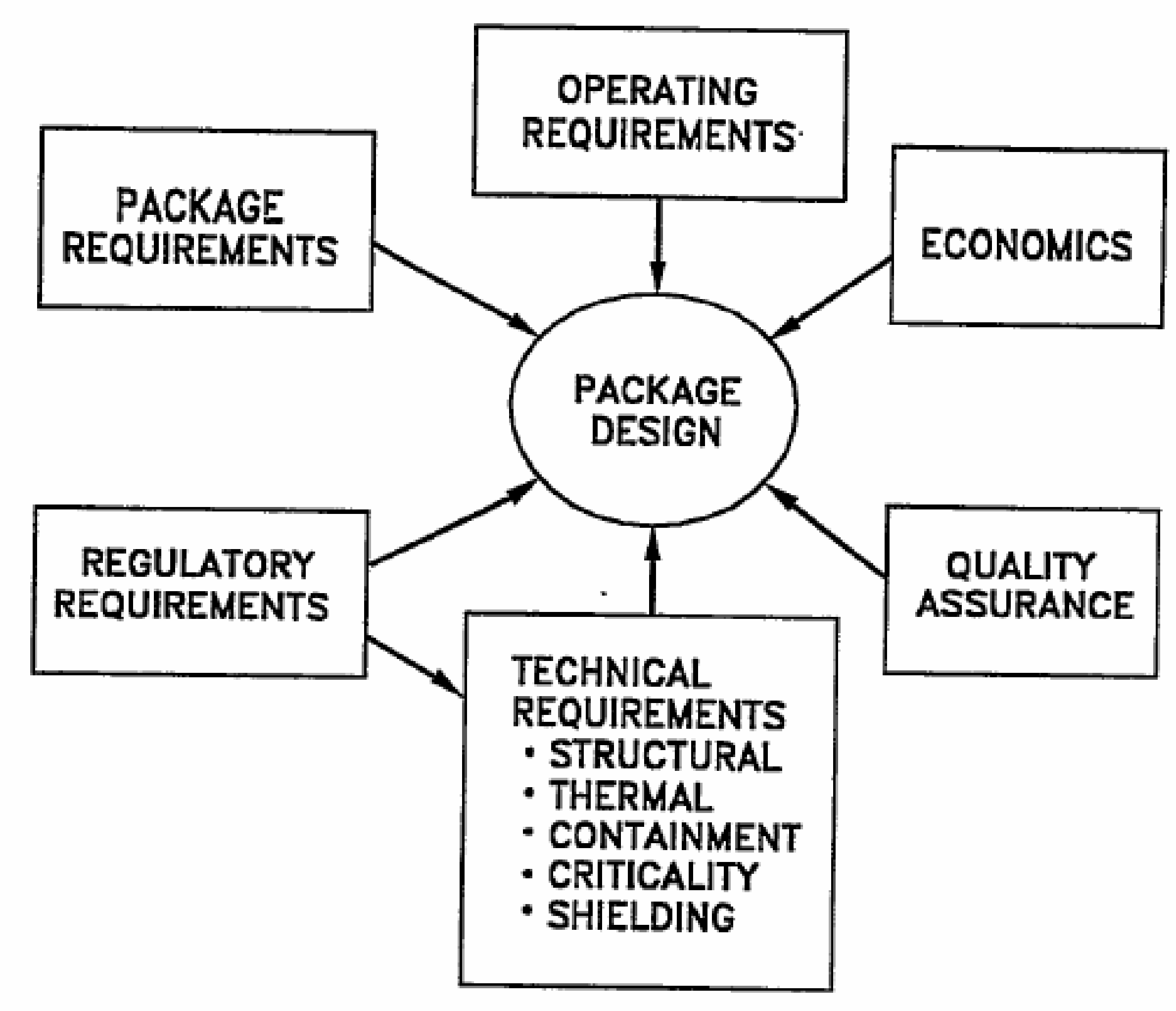
Licensing Basis

- Regulations for the Safe Transport of Radioactive Material (IAEA *Safety Standards Series No. TS-R-1*)
- Safety Analysis Report for the Packaging - SARP (according to USNRC Regulatory Guide 7.9, r1)
  - General Information (packaging description, drawings, QA)
  - Shielding Evaluation (gamma and neutron radiation)
  - Criticality Evaluation (criticality models for MTR and TRIGA fuels)
  - Operating Procedures (loading and unloading, dry/wet (un)loading)

Licensing Basis

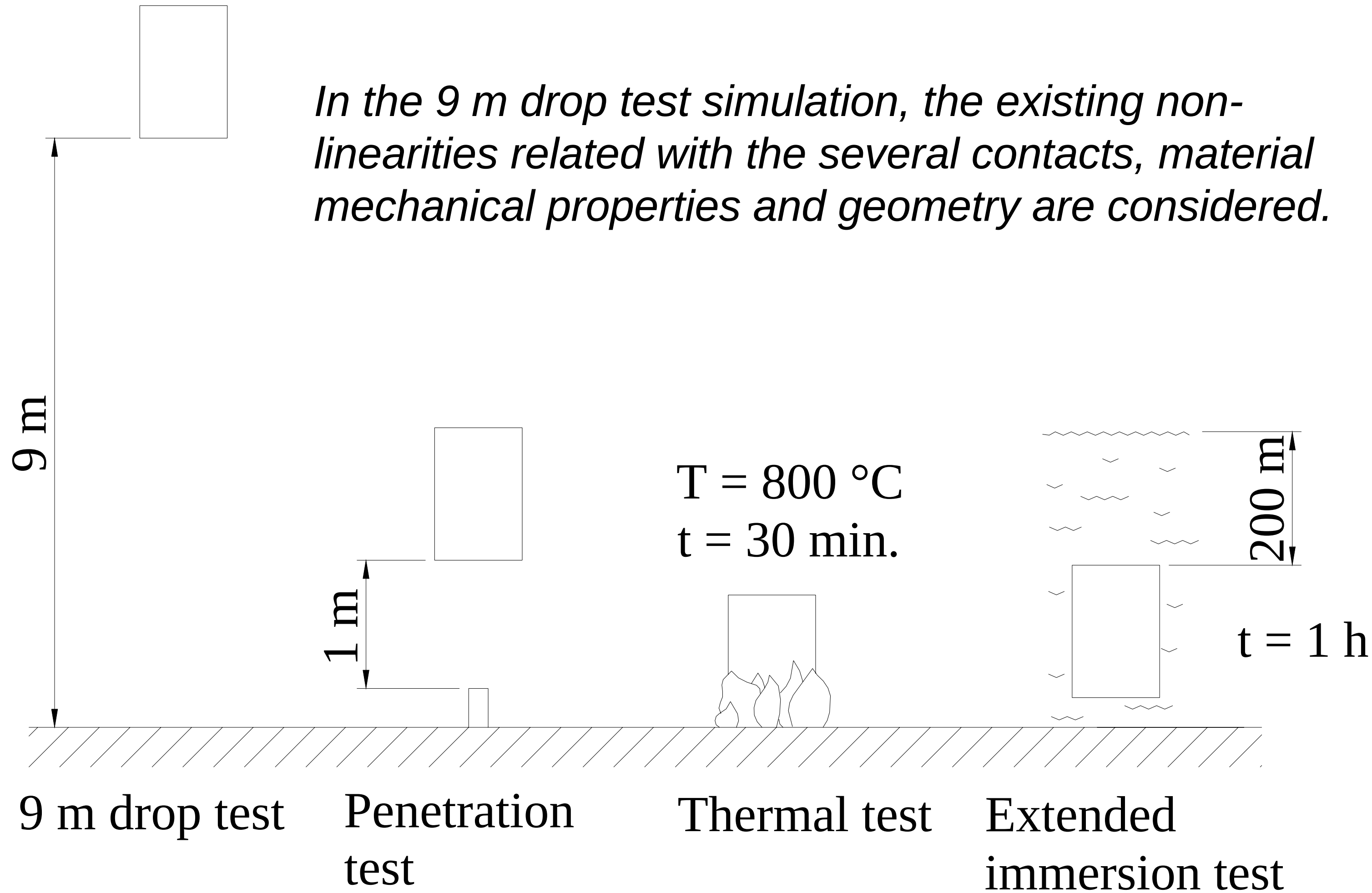
- Regulations for the Safe Transport of Radioactive Material (IAEA *Safety Standards Series No. TS-R-1*)
- Safety Analysis Report for the Packaging - SARP (according to USNRC Regulatory Guide 7.9, r1)
  - Structural Evaluation (materials, lifting and tiedown devices, normal and accident conditions)
  - Thermal Evaluation (thermal properties of materials, normal and accident conditions)
  - Containment (containment boundary, normal and accident conditions)
  - Acceptance Tests and Maintenance Program (visual inspection; structural, thermal and leak tests; shielding integrity verification)

Licensing Basis



Prescribed tests

In the 9 m drop test simulation, the existing non-linearities related with the several contacts, material mechanical properties and geometry are considered.



FEA Objectives

- Demonstration of cask structural performance
- Use of scale models – testing facilities limitations and cost reduction
- To verify that worst cases have been chosen for drop testing
- To estimate strains and acceleration at prescribed positions (e.g., the fuel elements)
- Improvement of the cask structural design
- Development of models for future benchmarking and analyses
- Demonstration that the analysis model is sufficiently robust for use in future safety analyses

Scale Models

- The physical testing of packages can be expensive – destructive tests
- Small packages are more frequently tested than larger ones – the cost of testing is often not as great as it is in the case of large packages (the package itself is less expensive and the tests are more simple, in general).
- In the case of large packages – scale models can be used
- Models should not be smaller than a quarter scale
- Models should have sufficient detail to ensure that all important structural features are represented

Scaling Laws

Velocity	V_m	=	V_p
Weight	W_m	=	$S_m^{\ 3} W_p$
Deformation	d_m	=	$S_m d_p$
Applied force	F_m	=	$S_m^{\ 2} F_p$
Acceleration	A_m	=	A_p / S_m
Duration	t_m	=	$S_m t_p$
Momentum	MV_m	=	$S_m^{\ 3} MV_p$

Physical testing and numerical simulation interaction

- Model (1:2)
  - 9 m drop test + penetration test + thermal test
  - It is important to notice that the model thermal tests will be used to validate the thermal numerical simulations (model thermal tests are not acceptable in the cask qualification)
  - Damaged model
  - Extended immersion test (200 m) numerically simulated using damaged model

- Prototype: All tests numerically simulated

Cask design

- Design criteria: 21 MTR or 78 TRIGA, max. weight 10 t, Type B fissile package

- Design hypothesis:

  . 125 g in the internal basket (i.e., 250 g in the model 1:2)

- Main parts:

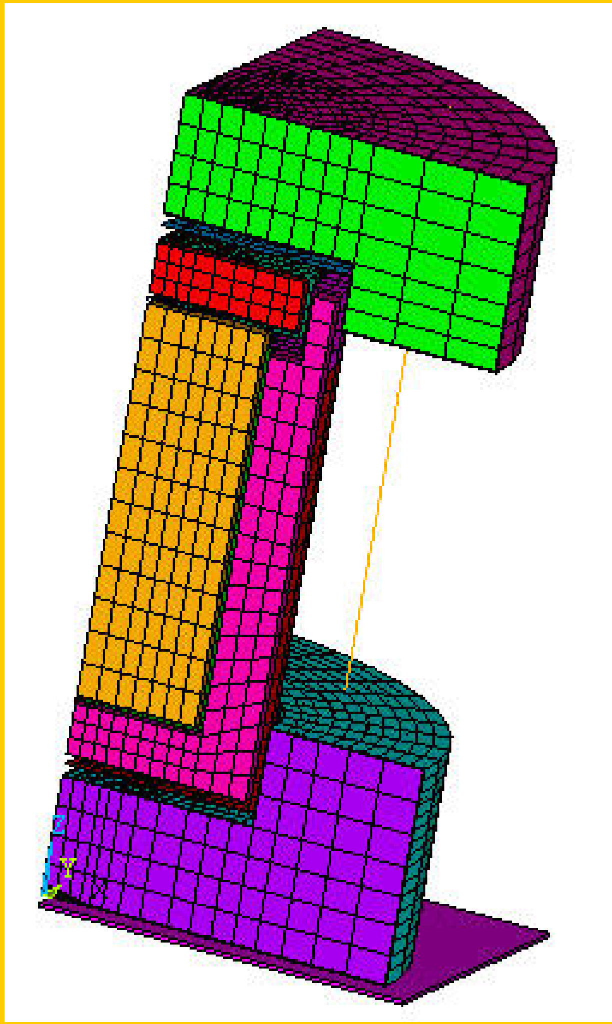
  . Main body; . Lids: internal and external

  . Basket; . Impact limiters

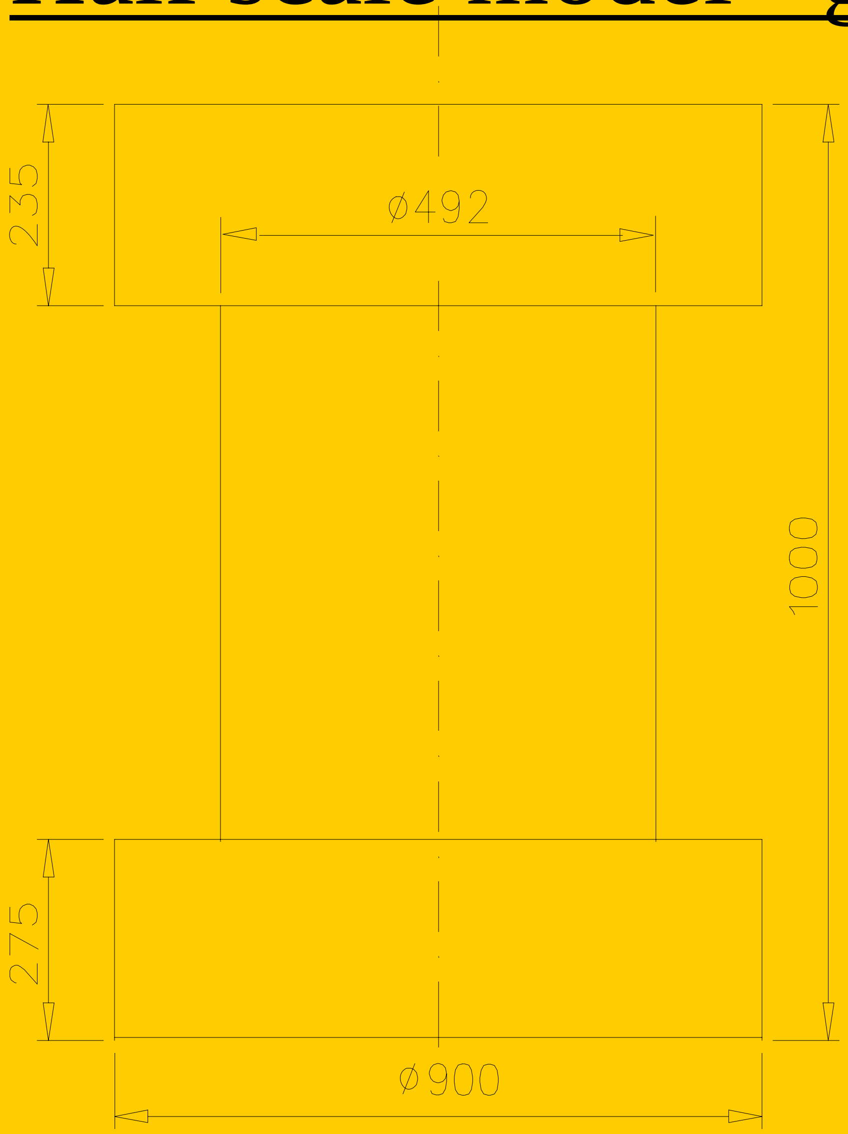
Cask model description

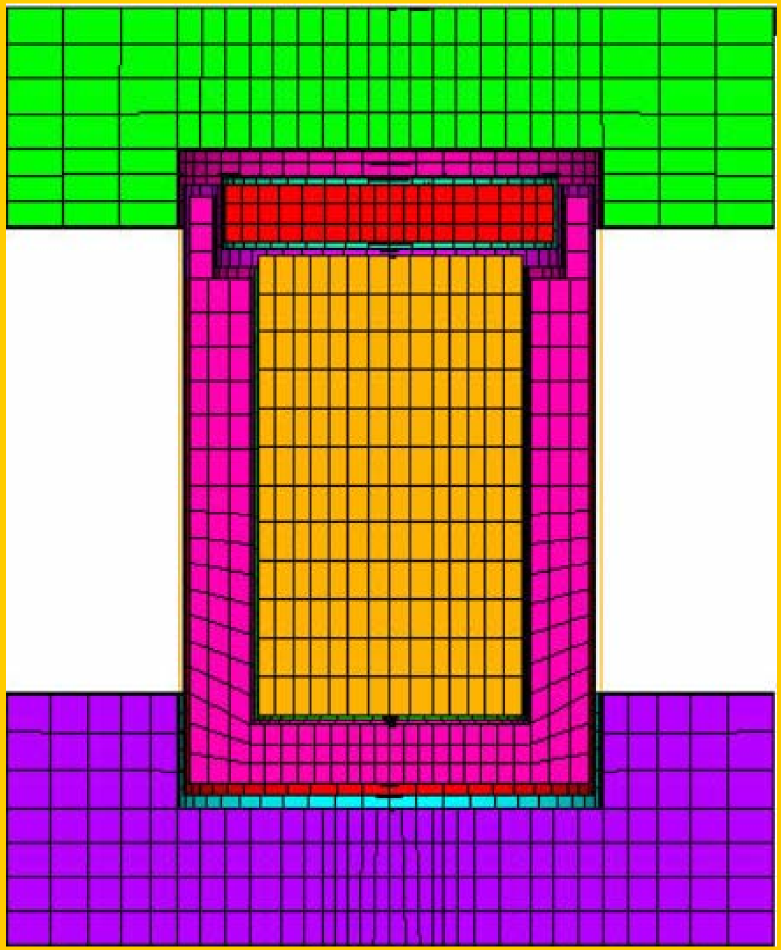
The cask itself is a stainless steel cylinder with flat heads (the bottom one is welded and the upper one has flanges with threaded connections) and internals (basket + FEs)

It is surrounded by lead shield and it has also upper and bottom wood dampers. All contained in stainless steel shells.



Half scale model - general view

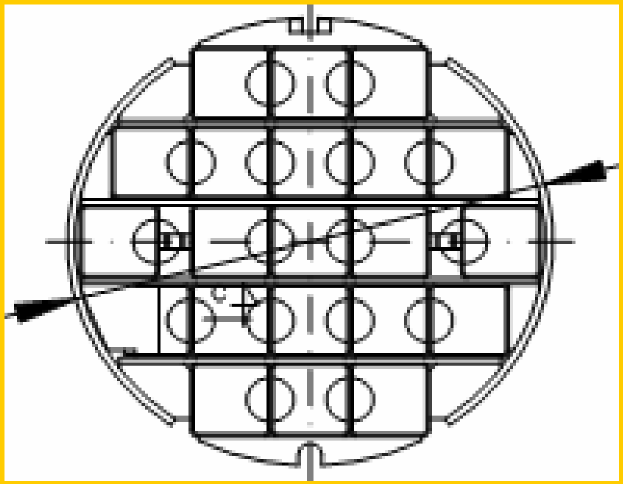




Main body

External wall: resist puncture
Internal wall: resist lead contraction

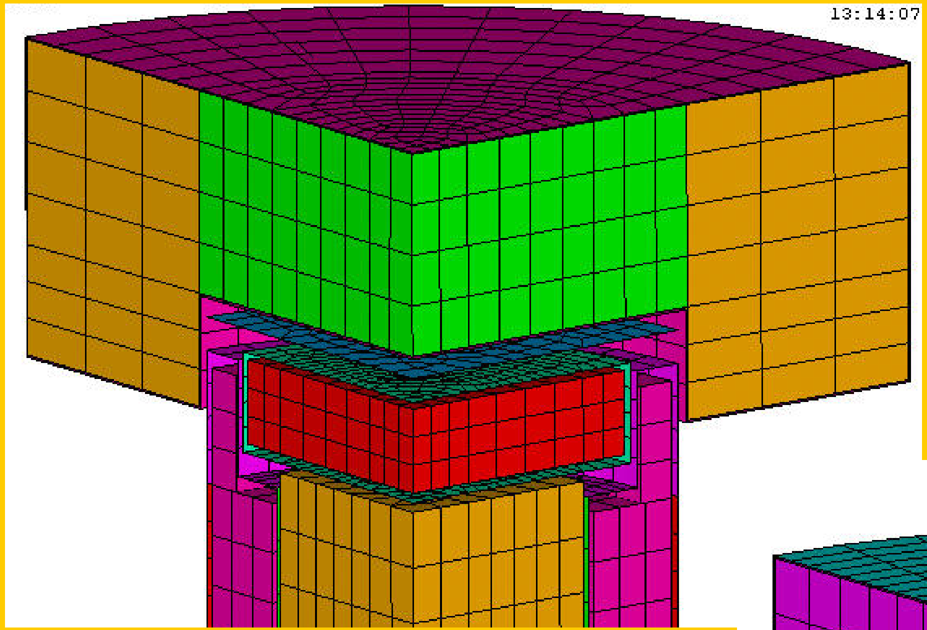
Internal basket (dummy mass in the model)

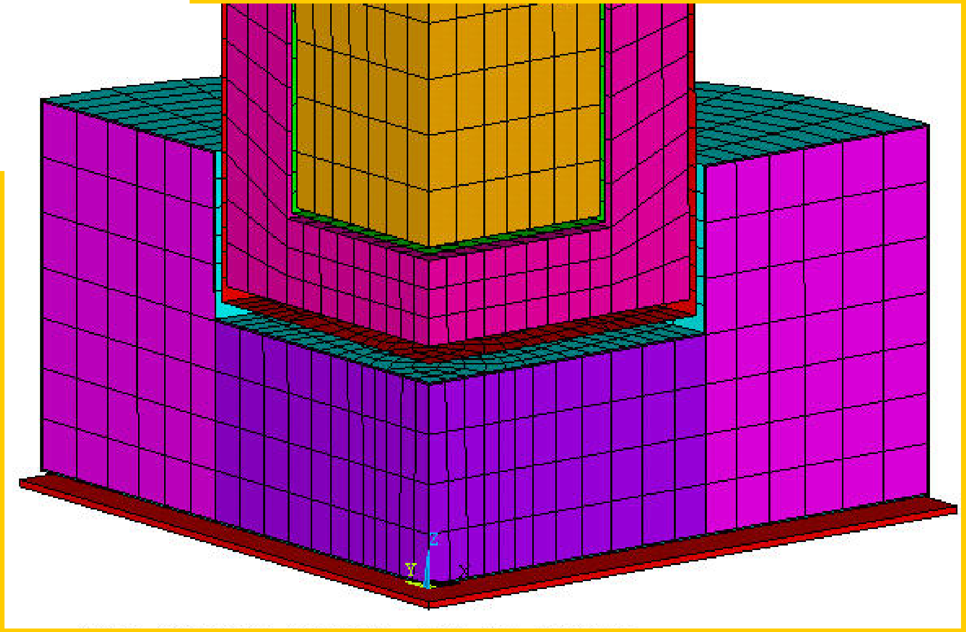


(1st approach)

- Only for MTR fuel elements
- 20 positions for fuel elements, one position for accelerometer

Impact limiters





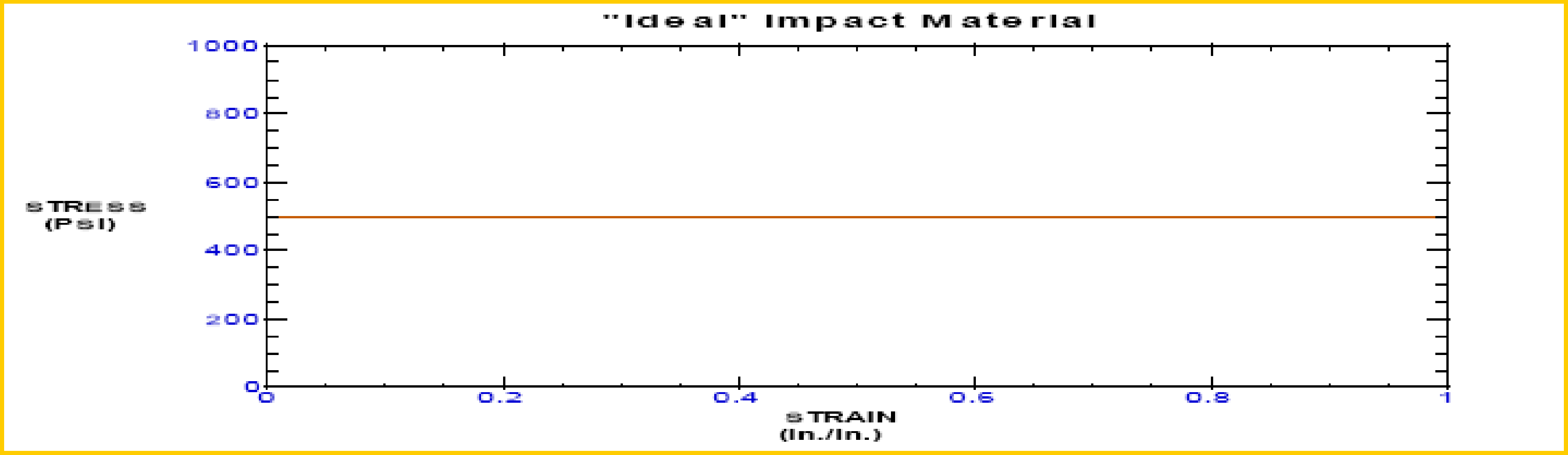
- Filling material:
  wood (OSB)

Impact limiters materials

Filling material	Advantages	Disadvantages
Solid wood (Eucaliptus, Pinus)	•High energy absorption capacity (tenacity)	•Flammable •Hard to model •Anisotropic, not homogeneous
Reconstituted wood (OSB)	•Homogeneous	•Flammable •Hard to model •Anisotropic (slightly)
Polyurethane foam	•Only slightly anisotropic •Easily modeled	•Challenging manufacture •Organic, flammable
Light mortar	•Homogeneous, isotropic •Good resistance to fire •Inorganic •Easily modeled	•Needs baking

Impact Limiters – Ideal Impact Behavior

- What would constitute the ideal deceleration for impact limiters?
- An ideal impact limiter would decelerate the payload uniformly throughout the available distance.

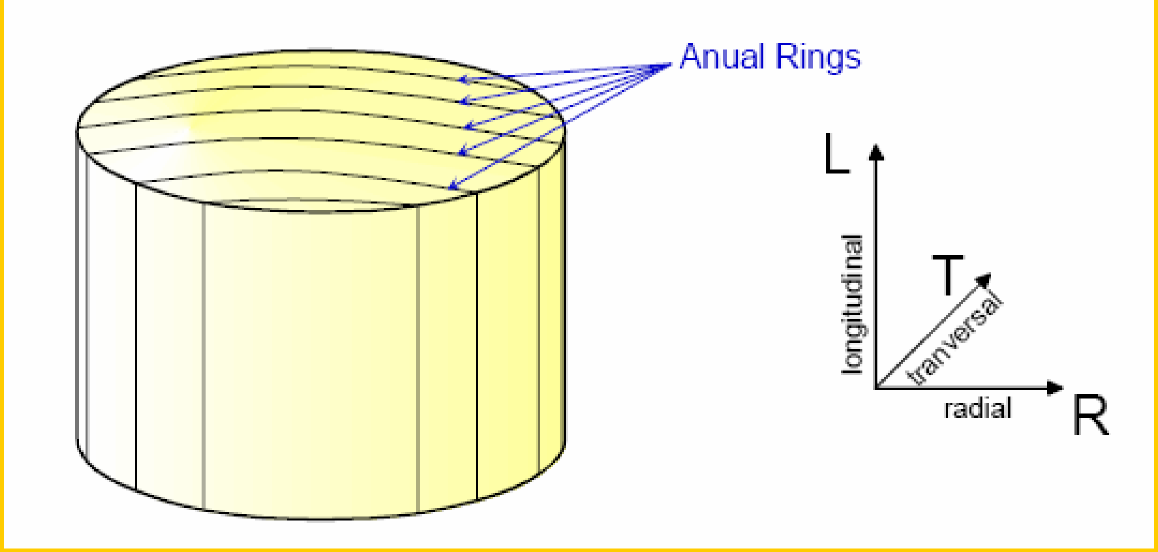


Numerical Simulation of Wood Filled Impact Limiter with LS-DYNA

- The material wood is used to absorb the kinetic energy
- Because of the specific compression behavior of wood, a special material model has to be applied for the numerical simulation.
- This model has to be capable to describe the non-linear deformation behavior taking into account the direction of fiber as well as the decrease of the volume.
- The program LS-DYNA provides several material models designed for foams or honeycomb structures that are able to describe compressible behavior.
- For the application of these models to wood, suitable material parameters have been determined on the basis of simulations of experiments with cylindrical specimens.
- The determined material parameters have been applied to the simulation of a drop tests of a transport and storage cask.

Numerical Simulation of Wood Filled Impact Limiter with LS-DYNA

- Wood may show significant anisotropic deformation behavior at elastic and plastic compression. This is resulting from its cellular microstructure where the aspect ratio of the cells is about 25. The cells are aligned with their long axis in the direction of the trunk. Therefore the deformation behavior along the long axis of the cells (longitudinal L) is different from the behavior of the deformation of the cells perpendicular to this direction. Whereby the difference behavior between radial (R) and tangential (T) deformation is relative small. The material could be described as transversal-isotropic.



Numerical Simulation of Wood Filled Impact Limiter with LS-DYNA

- Plastic material models for finite element simulation need to have a yield criterion a flow rule and strain softening/hardening law.
- The yield criterion defines a surface in the multi-axial stress domain, which separates stress states leading to elastic and non-elastic deformation.
- The flow rule defines the direction of the inelastic deformation and the strain softening/hardening law defines the movement of the yield surface as a consequence of plastic deformation or densification.
- For wood densification no specific material model was found which is able to describe the non-linear transversal deformation process with all details; however there are some approaches to model the deformation behavior of cellular material, which may be applicable to wood.

Numerical Simulation of Wood Filled Impact Limiter with LS-DYNA

- Honeycomb type material models basing on the honeycomb approach are in general suitable for the description of orthotropic material, which shows extensive compressive behavior.
- The elastic behavior of the model before compaction is orthotropic where the components of the stress tensor are coupled. For honeycomb models the non-linear plastic behavior is modeled with load curves giving the stress strain relationships for normal and shear stress directions related to the material axis.
- Plastic behavior is uncoupled and as a consequence of this no Poisson´s ratio is considered.
- The honeycomb models are easy to use and adequately precise for stress states which are dominated by mono-axial compressing along the main material axis (in-axis loadings).

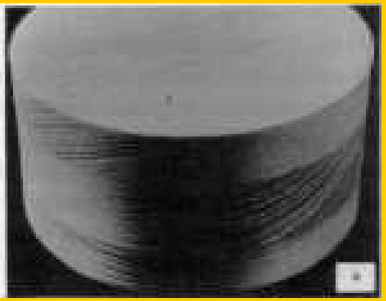
Numerical Simulation of Wood Filled Impact Limiter with LS-DYNA

- Foam type material models are based on material approach with defined yield surface and hardening/softening low in isotropic stress state and are able to model extensive compressive material behavior.

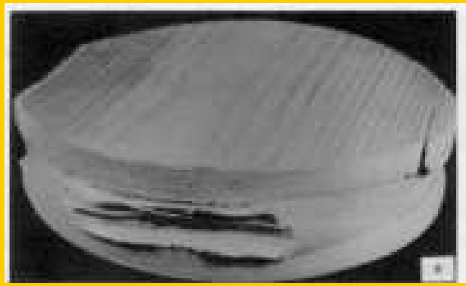
- LS-DYNA materials models options
  - Type 26 – MAT HONEYCOMB
  - Type 63 – MAT CRUSHABLE FOAM
  - Type 142 - *MAT TRANSVERSEL ANISOTROPIC CRUSHABLE FOAM
  - Type 57 - *MAT LOW DENSITY FOAM

Material Characterization

Non encapsulated and Encapsulated Specimens

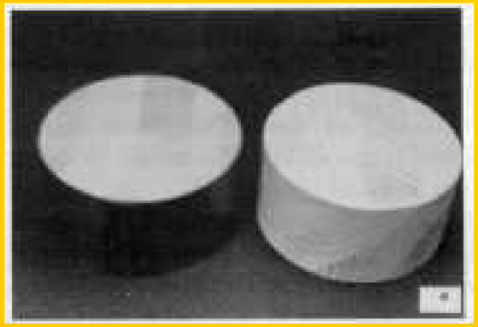


Based on the paper
R. Diersch, M. Weiss, G. Dreier, Investigation of the impact behaviour of wooden impact limiters, Nuclear Engineering and Design, 150(1994), p. 341-348



grains perpendicular to the load (end-on impact)

grains parallel to the load

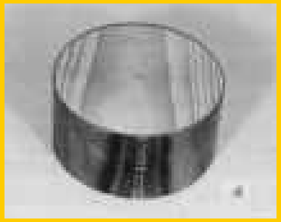


grains inclined at 20 °, 45 ° and 70 ° to the load (edge impact)

grains 50% parallel, 50% perpendicular to the load (side impact)

Material Characterization

Non encapsulated and Encapsulated Specimens



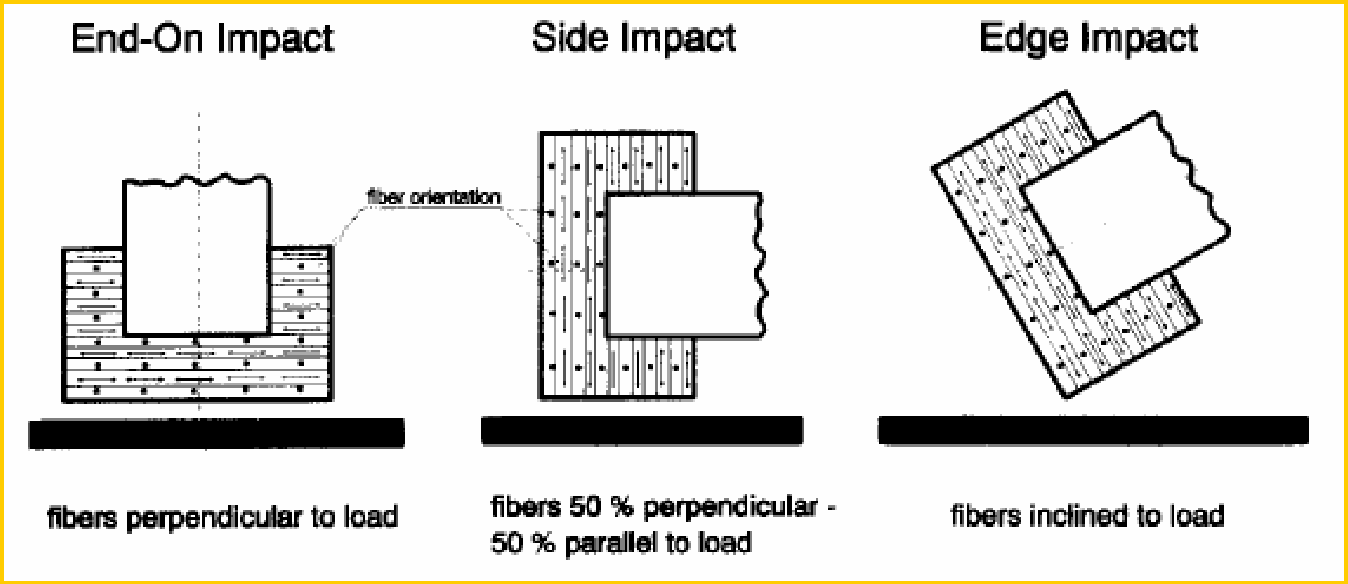
Based on the paper
R. Diersch, M. Weiss, G. Dreier, Investigation of the impact behaviour of wooden impact limiters, Nuclear Engineering and Design, 150(1994), p. 341-348

grains perpendicular to the load with steel casings of 0.5 mm and 1 mm

Material Characterization

Encapsulated Specimens

Based on the paper
R. Diersch, M. Weiss, G. Dreier, Investigation of the impact behaviour of wooden impact limiters, Nuclear Engineering and Design, 150(1994), p. 341-348



Material Characterization

Encapsulated Specimens

Based on

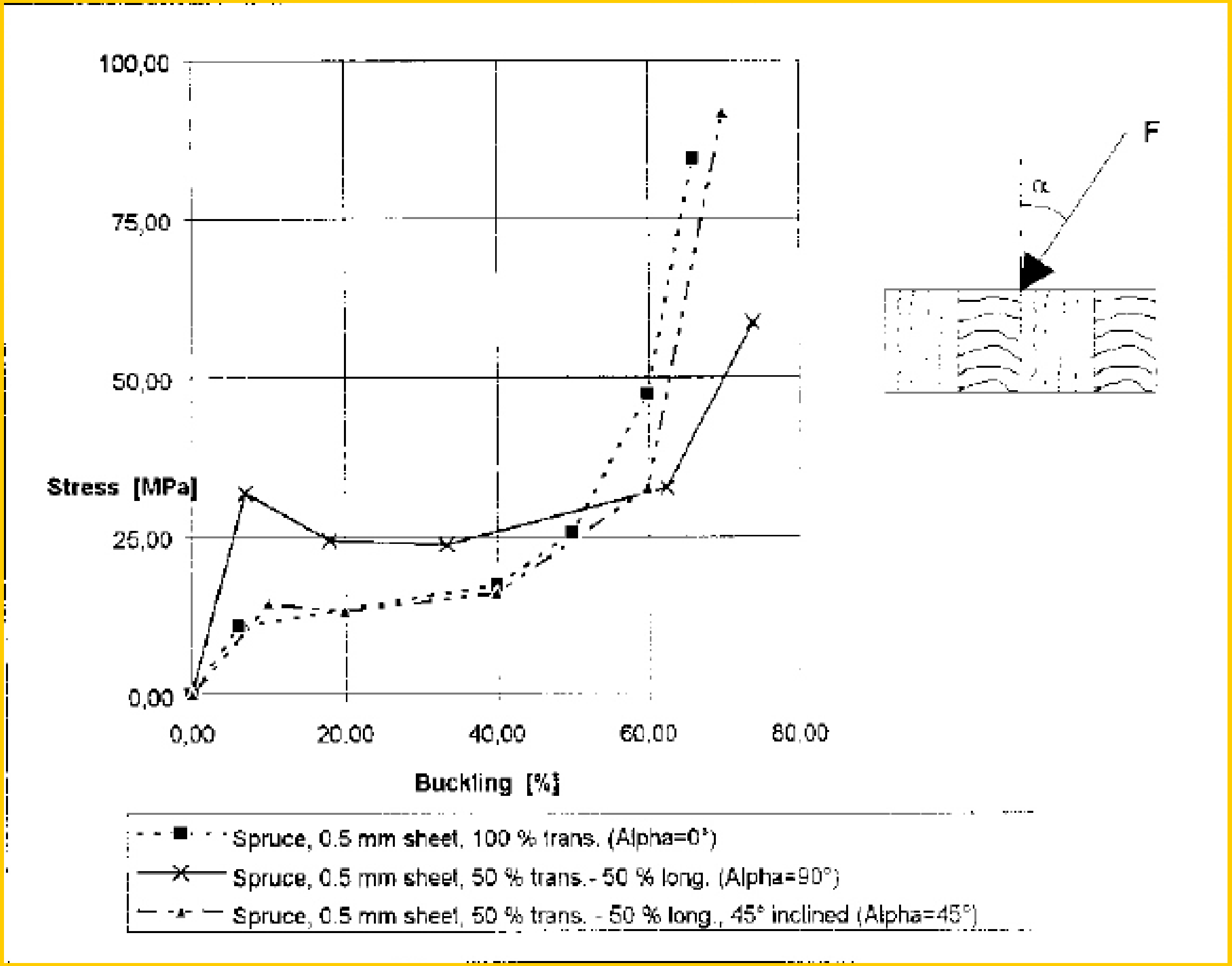
CONSTOR® Full Scale Drop

Testing Program

NRC Meeting

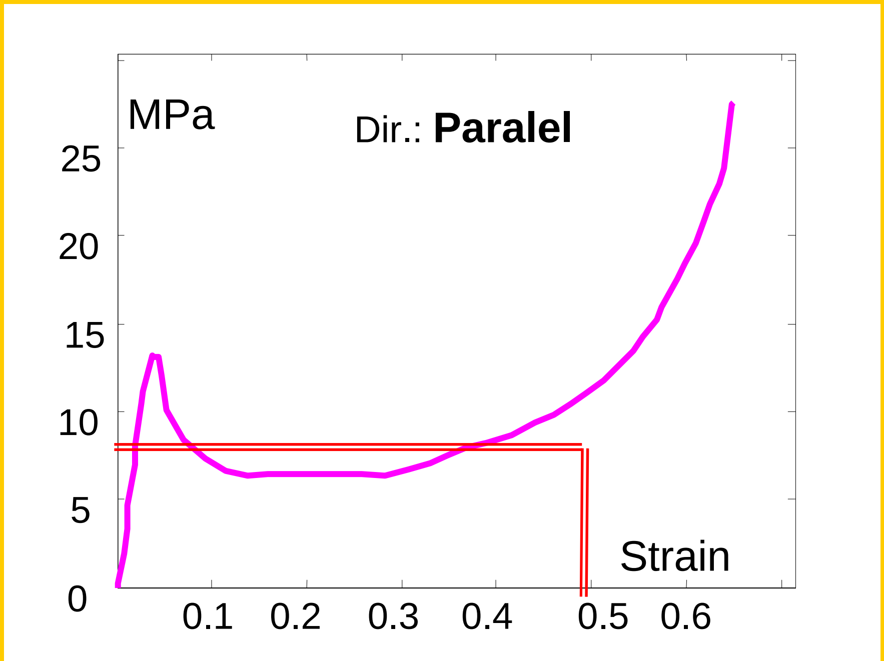
September 1, 2004

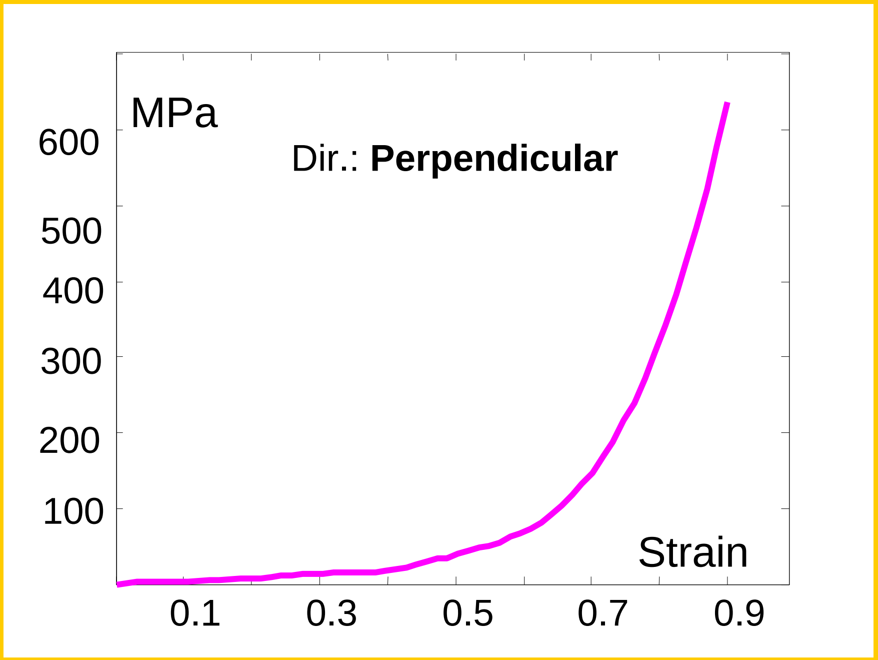
Part 2



Impact limiters – Material Properties

Material: Oriented Strand Board





- OSB properties – dynamic, non encapsulated, no inclined direction
- Recommendations to develop new dynamic OSB characterization tests using encapsulated specimens and testing in inclined directions

Items to Be Verified

Cask body (single or multiwall)
Closure lid(s)
Valve covers for containment penetrations
Basket for fuel or radioactive material
Impact-limiters (or shock-absorbing structure)
Impact-limiter attachments
Lifting/rotation trunnions
Cask support structure and tiedowns
Transport vehicle (which determines normal operation loads)

Analysis Data

Cask Part	Material	Dimensions
Lower shell	stainless steel	dia = 900 mm
Lower Damper	wood (**OSB**)	dia = 894 mm
Inner Shell	stainless steel	dia = 328 mm
Lead	..	
Outer Shell	stainless steel	dia = 492 mm
Upper Damper	wood (**OSB**)	dia = 894 mm
Upper shell	stainless steel	dia = 900 mm
Tie bar	stainless steel	dia = 30 mm

Contacts, Materials & Loading

1. Contacts – defined as ASTS in the ANSYS LS-DYNA

(Automatic Surface-To-Surface Contact)

2. All materials, but the OSB and the rigid surface, were modeled as Bilinear Isotropic Material (BISO)

The rigid surface was modeled with the RIGID option and same properties as the steel

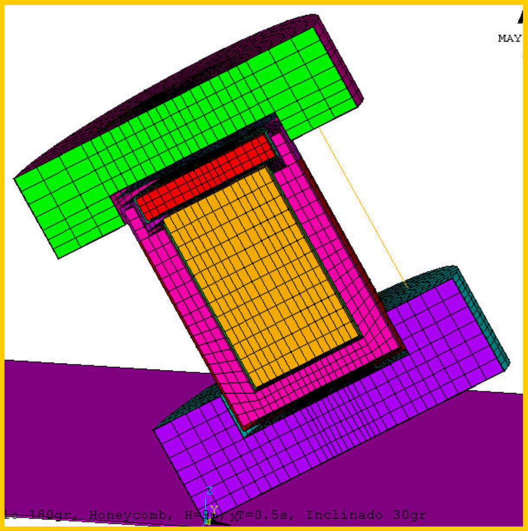
3. Loading - **initial velocity** (corresponds to 9 m drop plus the **gravity acceleration**

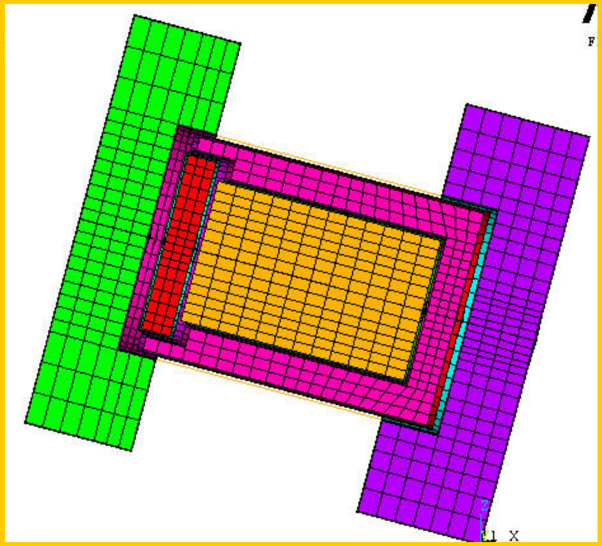
<u>Analysis done and future analyses</u>

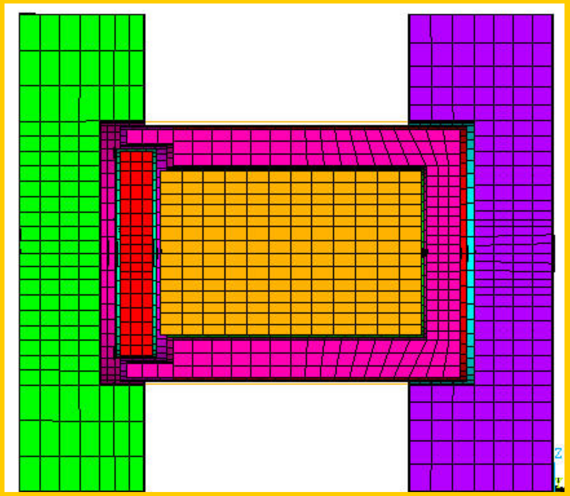
. The most damaging position should be investigated

. The first one analyzed was the upright one (90º FE Model)

. Some skewed positions will also be analyzed







Only the upright drop results will be presented

Results (90º FE Model)

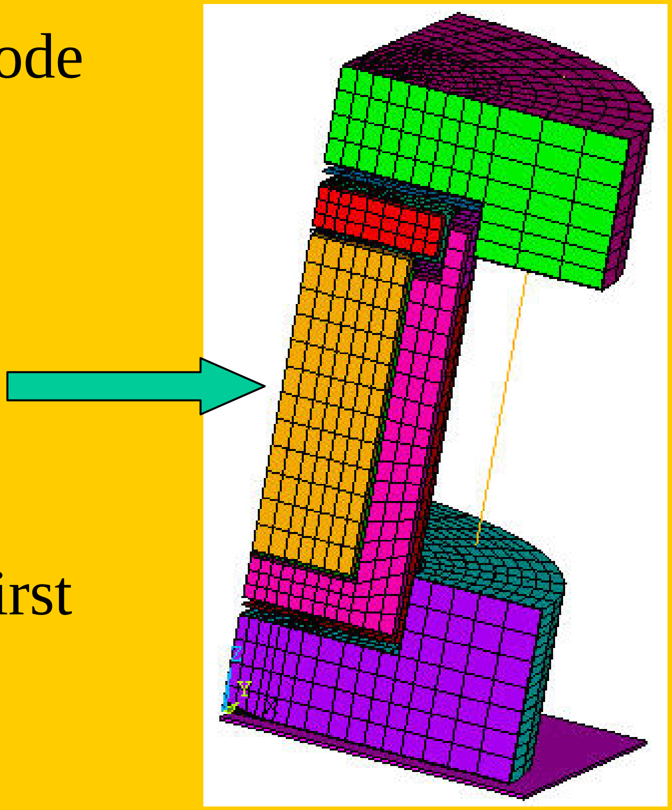
Results presented for the indicated node (in the middle of the internal mass):

• the vertical displacement

• the vertical velocity

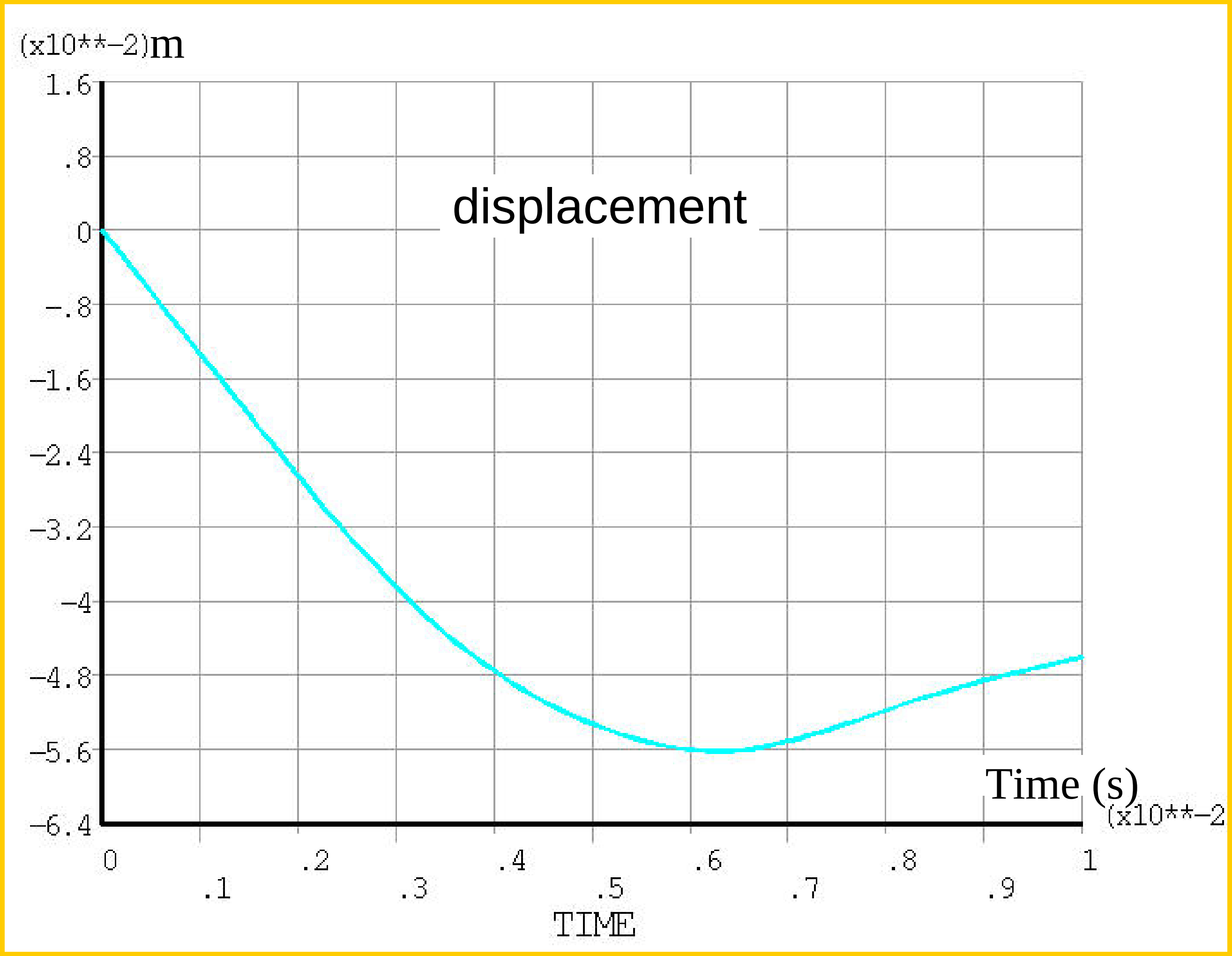
• the vertical acceleration

• the deformed structure during the first 10 ms of the analysis



The analysis was performed until 20 ms

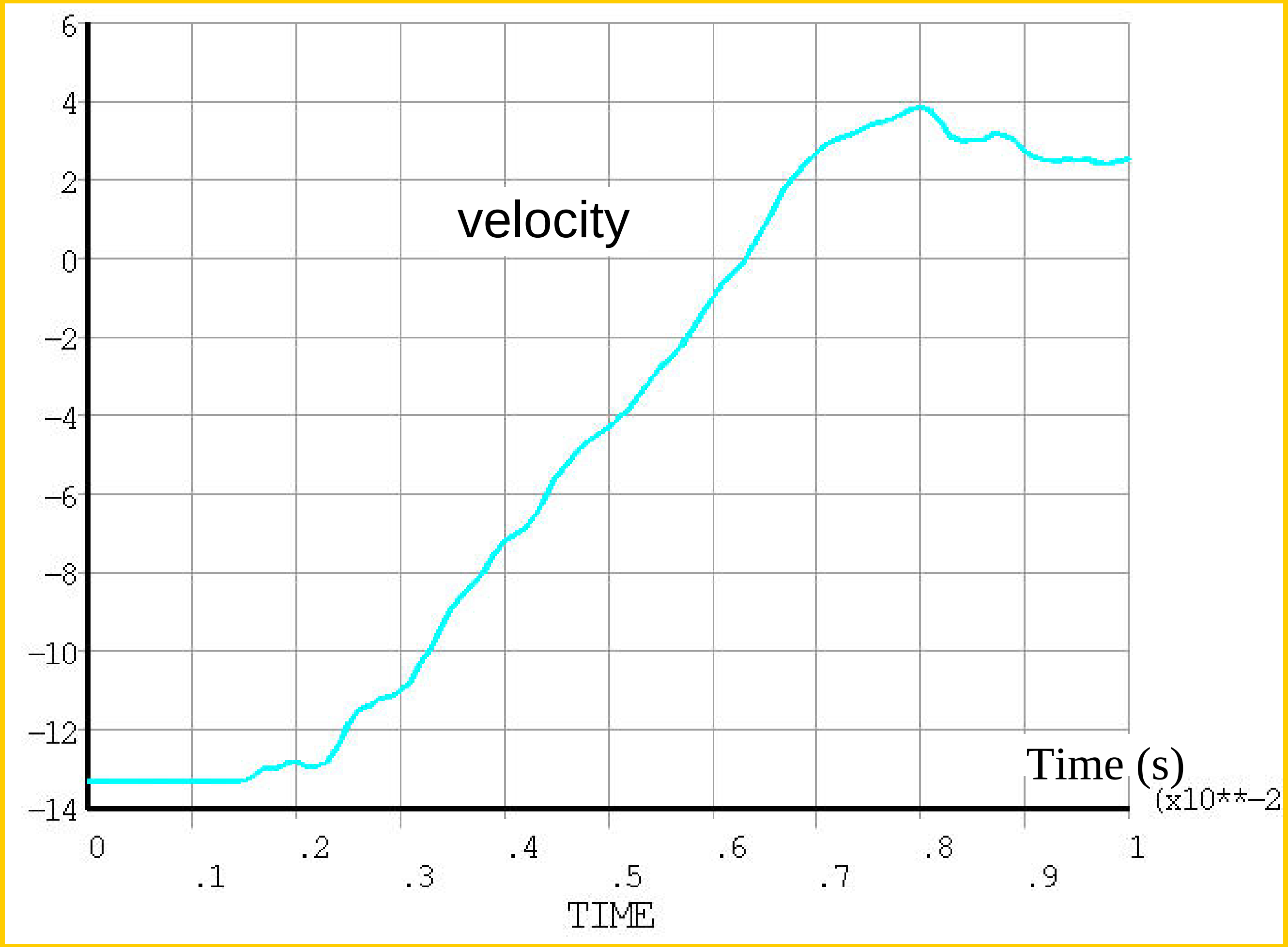
Results



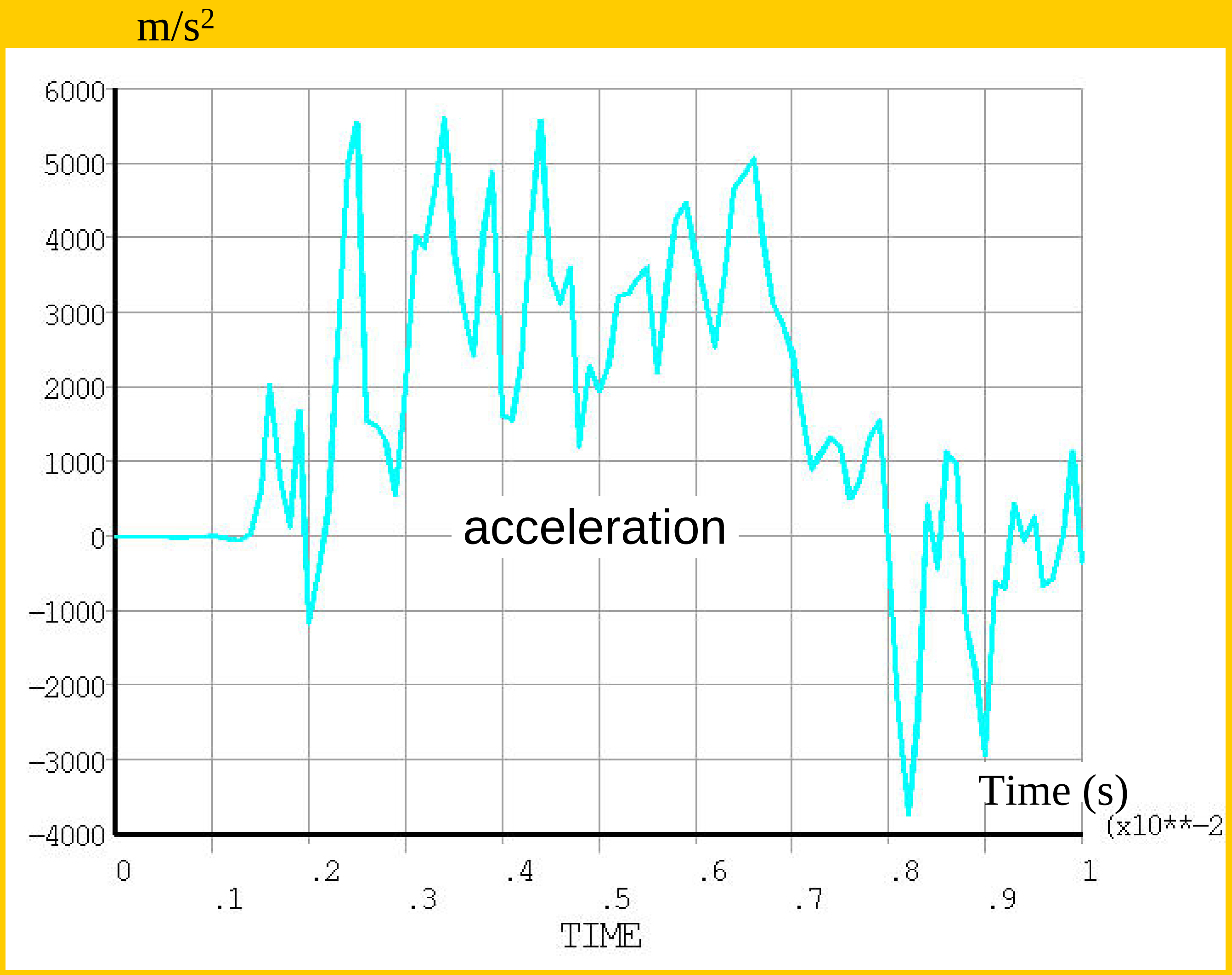
(x10**-2)m
1.6
.8
0
-.8
-1.6
-2.4
-3.2
-4
-4.8
-5.6
-6.4
displacement
Time (s)
(x10**-2
0
.1
.2
.3
.4
.5
.6
.7
.8
.9
1
TIME

Results

m/s



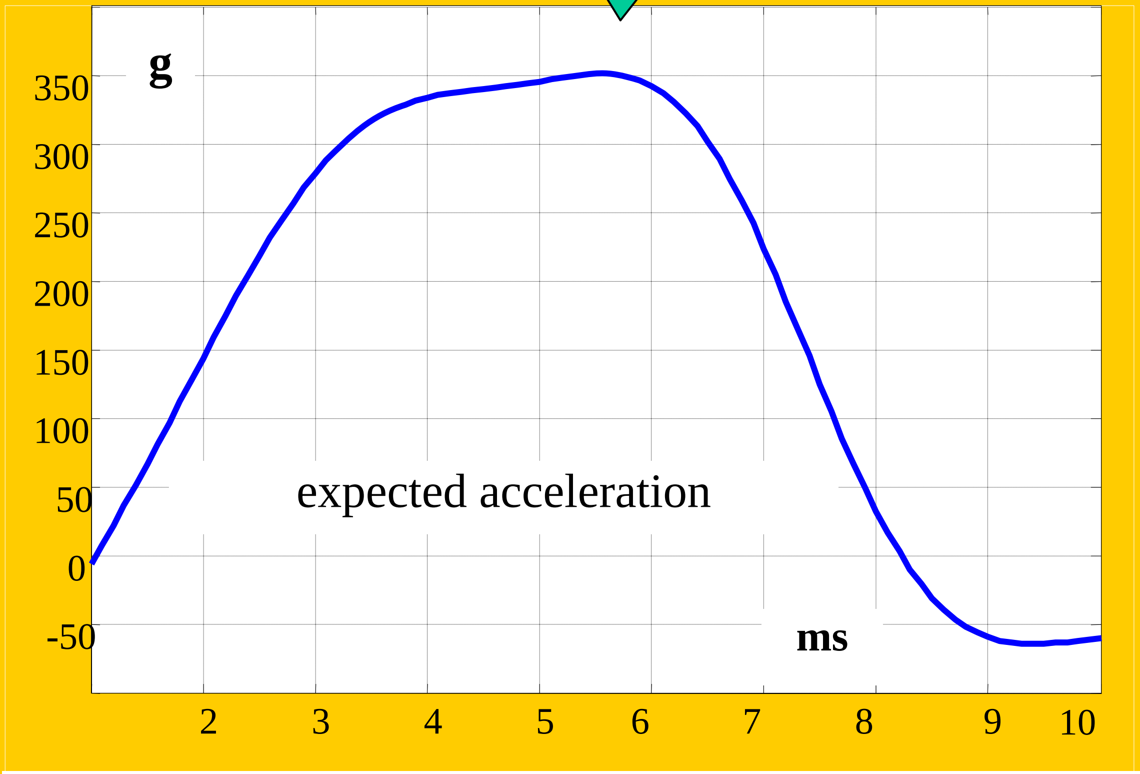
Results

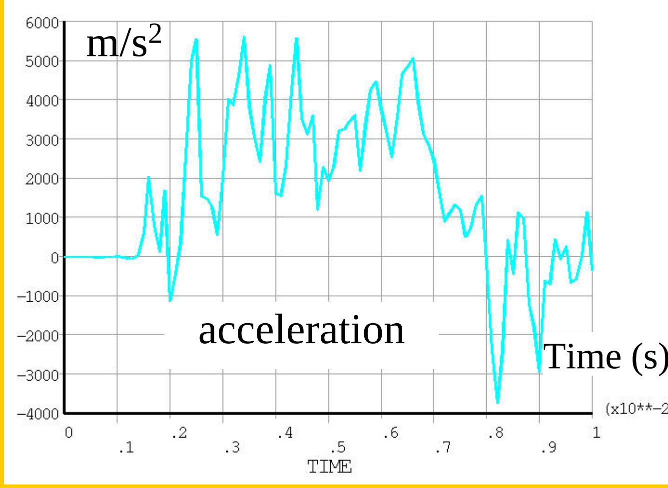


m/s2
6000
5000
4000
3000
2000
1000
0
−1000
−2000
−3000
−4000
acceleration
Time (s)
(x10**−2
0
.1
.2
.3
.4
.5
.6
.7
.8
.9
1
TIME

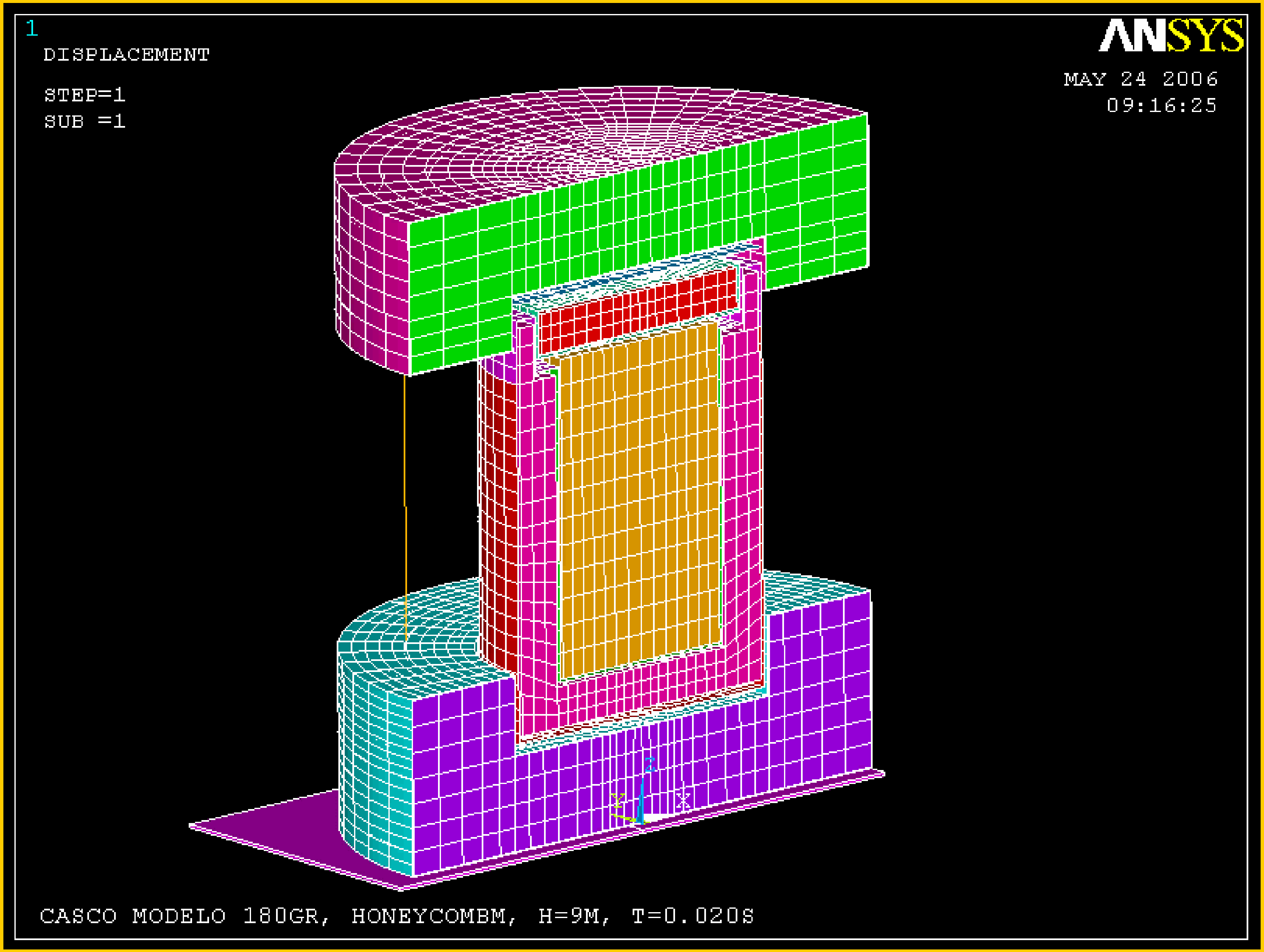
Results

Max Acceler.
≈ 350 g





Acceleration – After Signal Filtering


1
DISPLACEMENT
STEP=1
SUB =1
ANSYS
MAY 24 2006
09:16:25
CASCO MODELO 180GR, HONEYCOMBM, H=9M, T=0.020S

Discussion and Conclusions

- This is a first analysis and it was performed with preliminary data (mostly for the materials and, in some aspects, for modeling)

- Scale correction: some values measured in the model or obtained during the model testing should not be directly associated with the full cask. In special, for the acceleration:

$$Acc_{moodel} = \frac{Acc_{protot}}{S_f}$$

Half scale ➔ $S_f = 2$

- The obtained max. accel. (≈350 g) in the model internal mass gives ≈175 g in the prototype which is compatible with the project design hypothesis (≈125 g)

Discussion and Conclusions

•For the analyses in skewed positions the FE model will be doubled (180º) with symmetries defined accordingly

•The model for the final simulations will be calibrated with experimental data

Once the model is calibrated, the prototype structural qualification will be done numerically

Next Steps

- Improvements in the modeling – recommendations from the literature and from K. Handy (ORNL & NTRC)
- More refined finite element models
- Processing difficulties – needs for improved solution strategies and powerful hardware
- Recommendations for the hardware configurations

Modeling Improvements

- Mesh coarseness/fineness appropriate for purpose of analysis
- Mesh refined at areas of higher stress gradients and areas of larger deformation
- Mesh coarsened at areas of lower stress gradients and deformation gradients
- Mesh refined where higher accuracy required
- Identical mesh for identical components in similar loading
- Identical mesh of similar or repeating geometry
- Aspect ratio not exceed 2:1 to 3:1

Recommendations from K. Handy (ORNL & NTRC)

- **1. Is the discretization (mesh) adequate (it has about 6000 elements and about 8000 nodes)?**
- It all depends on what you are modeling, and the response you want to obtain from your model. If you are looking for global response of the container, the element mesh you have may be sufficient. However, if there is a question about the integrity of the tie rods and their connection to the package, then much more detail is required. If the connection of the tie rods to the package is of interest/concern, then this connection needs to be modeled explicitly. Including any welds, nuts, washers, etc. I have found that even components which seem insignificant, may force a different response than if they are not there (e.g., washers), therefore due to the path dependency of the problem, they may be very important.
- Critical welds should be explicitly modeled. I generally use a one element row to model fillet welds. Full penetration welds, I model by allowing the opposing component meshes to merge at the weld.
- Critical bolting connections are explicitly modeled. They include the shank, the nut and any washers. The shank at the nut should be modeled such that the faceted area of the shank equals the tensile stress area of the threaded region. I generally allow the nut mesh to merge with the shank. In bolted
- connections, the weak link is the shank, not the threads. In short studs, the little extra length of the shank due to the washer, could provide relatively significant extra length for energy absorption in tension.
- It appears the stacked OSB is modeled as a homogeneous solid. This brings up the question of shear capability between the sheets. Are the sheets glued together such that the sheets do act together? This concern does not matter much in a pure end impact. But in a side impact, some separation could occur, even though that would be minor. The concern would be most evident in a corner impact. It does not appear that the current model would model possible shearing of the sheets in a corner impact. Shearing between sheets would change the response of the overall container. Is this significant? You need to ask yourself that question based on what you are trying to obtain from the model.
- Generally, my models are on the order of 1e5 to 5e5 elements, but that is simply due to the level of response I want to obtain. I run on dual processor PCs and a typical impact solution will last about a day for a 0.015 second solution.

Recommendations from K. Handy (ORNL & NTRC)

- **2. The contact definition (ASTS) is adequate for this analyzed situation or should an other one be used instead?**
- I assume the ASTS is referring to “automatic surface to surface” contact. I have not used that contact much. I generally use the “automatic single surface” or the TrueGrid defined “surface to surface” contacts. Generally, I
- have found the single surface contact to work well with material of similar stiffness. If there is a case of significant differences in stiffness (foams to stainless steel), I generally use the surface to surface contact at the outer surface of the softer material.
- If the ASTS appears to be doing its job, then I would say stay with it. You might try another contact (such as automatic single surface) to see if there is a significant difference to the response.

Recommendations from K. Handy (ORNL & NTRC)

- **3. Threaded connections. The threaded connections were not modeled although all contacts among the connected (parts) surfaces were. It is necessary to have them in the model? If affirmative, how can they be modeled? One simplified possibility is to couple the corresponding nodes and verify the forces in the coupling. Is that correct for this type of analysis?**
- This is somewhat a similar question to #1. It depends what you want to get from the model. If you want to know that a weld/threaded connection will not fail, then it must be explicitly modeled in detail.
- I do not believe that “One simplified possibility is to couple the corresponding nodes and verify the forces in the coupling. Is that correct for this type of analysis ?” is a good approach. If I deal with forces, I will think of statics and compare static loads against a static capacity. The nature of the Dyna problem is dynamic, or energy absorption capabilities. I think you need to think in terms of strain, or deflection not forces and stress. The dynamic problem is path dependent. It is very tedious to say that a model is “conservative”. If there is a threaded connection that is in question, you need to model it explicitly (as accurately as possible) and have Dyna handle the loads/stiffness/time relationship. I have found that it is best if something is modeled accurately. If the connections are of concern they need to be modeled in detail.
- The conservatism of a Dyna model is in its accuracy. In the static world, to make something thicker, or “beefier” is generally conservative. But in a dynamic world, thicker or beefier is more mass and a drastically changed response due to its altered mass/stiffness. It might be “better” in a static sense, but in a dynamic sense it may prove to alter the response such that you end up with a totally different response than you obtain by test. Making something stiffer or more flexible changes the response. So accuracy is the key in a path dependent analysis.

Recommendations from K. Handy (ORNL & NTRC)

- **4. Where (in what part of the model) should we take the numerical results (accelerations) to compare with the simplified calculation or with the experimental data?**
- **5. How these comparisons should be done: taking the acceleration of one single point ? Where and in which component? Averaging accelerations in a region (some points at the same level but including different components)? Where ? Averaging accelerations of a single component (some points at different levels) ? Which one?**
- I have taken the liberty of combining questions 4 and 5 because they are related.
- This is a question that should be well thought out. Everything in the model will effect the highly seemingly sporadic response of a single node. Material models, contact, the modeling technique (shells vs solids), material damping, and nodal/element spacing will all have an effect on the magnitude and ringing
- response. You would definitely want to choose a location which is somewhat locally stiffened. Not a sheet metal surface, but a thicker, locally reinforced location.
- If this is a critical concern, then you may want more of a one-to-one test/analysis comparison where you remove a lot of unknowns. Maybe a special test of a simple right cylinder being dropped on its end, with an
- accelerometer at the opposite end. Then model up the test and sense the nodal accelerations at a location similar to that used in the test. This way, you deal with fewer unknowns which can soon cascade and hide true reasons for discrepancies. This simple test would then form a basis, or a defense for how you filter or modify the nodal response for any review questions that might arise. I would then use similar accelerometers for the actual test specimen.

Recommendations from K. Handy (ORNL & NTRC)

- **6. Its was noticed a great difference among results (accelerations) along a radial position in the same component or across different components (the later seems quite obvious) as well along a vertical position. Is that a regular behavior or should it be a signal of some trouble in the model? If affirmative, is there any hint about it?**
- Yes, I have noticed somewhat similar results and no I don't think it is a signal of trouble. I do not deal with accelerations very often, I generally am interested in the filtered derivative of general displacements, or strains. Generally speaking the mesh near the center will be more square, or checker board and will transition to more concentric rows of elements near the outer radius. The effect you mention if quite obvious with a "butterfly" mesh as typically obtained with Ingrid, or TrueGrid. Model and investigate a thinwalled, simple right cylinder (like a paint can) in a pure end impact. The upper head will experience a response (plastic strains) which are seemingly not symmetrical where the mesh is projected and merged.
- My experience is that this is the "nature of the beast." If this is a concern with this response, my opinion would be to minimize any unsymmetrical aspects of the mesh pattern. This would include a much finer mesh and possible transitions in the concentric mesh (doubling or tripling the number of elements in the circumferential direction).

Hardware system configurations

- The available LS-DYNA license runs under Windows 32 OS and the parallel processing is not allowable. It is possible to migrate to Windows 64 or to LINUX 64 OS without additional cost.
- **General memory guidelines**:
  - The most general guideline for solver memory is **1 GB per million degrees of freedom**.
  - I/O requirements are the same as memory requirements for iterative solver jobs except that some analyses with multiple load steps may generate very large results files. Sparse solver runs require **10 GB per million DOFs for files**.
  - I/O cannot keep up with CPUs. This is a manageable problem with iterative solvers but it can result in severe performance degradation for sparse solver runs. Most modern operating systems, including Windows, now automatically use extra memory for a large file cache. **Large memory systems are therefore advantageous at every price point.**
  - High performance solution on any hardware configuration can be obtained when the model size is such that the memory requirement is comfortably satisfied within the available system memory.

Available hardware system configuration

- **Desktop 32-bit systems** (Windows 32-bit OS)
  - The desktop system has **2 GB of real memory. Swap space is also 2 GB.**
  - Disk resources is **250 GB of space**. This space is on a separate drive used as a scratch space when running large models, with permanent files moved to another drive or offline after the run completes. (Disk is SATA with 7,200 RPM)
  - Video card has **256 MB**.
- Windows 32-bit environment has a 2 GB limitation that reduces the conditions to process bigger models in an acceptable way.

Recommended hardware system configuration

- **Use of a desktop system with AMD Opteron and Intel em64t and Intel dual processors systems running 64-bit applications.**
  - Windows is released for 64-bit systems and the 2 GB limitation of the Windows32 environment need not constrain even commodity systems anymore. There may be some delay in fully implementing Windows64, depending on the speed of third party software and hardware suppliers in producing drivers and other software for Windows64.
  - As Windows64 is not supported by Microsoft in Brazil it is necessary to use Linux64 OS to avoid the 2 GB limitation.

Recommended hardware system configuration

- **Desktop 64 bits systems**
  - These systems should have a desirable **8 GB of memory and 2 processors**. Disks should be **10k or 15K RPM drives with at least 200 GB** of free space. The jobs should run in a separate, striped partition. 8 GB will allow sparse solver jobs of 500k to 750k to run in-core or very efficiently out-of-core. Most sparse solver jobs, even up to 1 million or 2 million DOFs will be able to run in an efficient out-of-core implementation that will not degrade performance like 32-bit OS experience. This is a sweet spot for these systems over 32-bit machines. An additional benefit of 8 GB of memory for smaller systems is much better elapsed time performance for those jobs that nearly fill memory on a 32-bit machine. Even jobs with 300-500 k DOFs may run entirely in-core, allowing the system to run at sustained peak performance.
  - Most of these systems still lag in delivering good sustained I/O performance. It is **best to run jobs whose memory usage is well within the 8 GB limit, thus avoiding the most costly I/O.**
  - These systems can use 2 processors effectively but they generally **should be considered single job/single user resources**. They generally do not have the OS refinements nor resources to support multiple users or more than one demanding job at a time.

Recommended hardware system configuration

- Motherboard - ASUS or INTEL
- Processor - Intel® Core™2 Extreme QX6800 (quad-core)
- Memory - 4 x 2 GB DDR2-1066
- Video card - ATI or NVIDIA 1 GB
- Hard disk 1 – 150 GB SATA or SAS 10,000 to 15,000 RPM
- Hard disk 2 – 250 GB SATA 7,200 RPM
- OS System – LINUX64