Session H2.2

CAPTURING THE BEST PRACTICES FOR THE ORGANIZATION AND ANALYSIS OF CRASH-TEST SIMULATION LOAD-CASES

Irene Makropoulou, Konstantinos Kiouptsidis, Konstantinos Skolarikis
BETA CAE Systems S.A., Greece

KEYWORDS - Simulation model robustness, process repeatability, crash-test simulation, Data Management, Task Manager

ABSTRACT - The current tendencies of modern automotive industry impose the reduction of the CAE turnaround cycle, promoting at the same time the requirement for the generation of robust simulation models. Currently, slight changes in model parameters may result in a great scatter in simulation results. The robustness of the simulation models also suffers from the incorporation of changes that originate from the product evolution process. To develop accurate models, the CAE engineers have a great necessity for pre- and post-processing tools able to safeguard and capture the best practices used for the simulation model build-up and evaluation, reducing the error-prone procedures involved in the CAE-cycle while ensuring the repeatability and versatility of the complete process.

This paper will present the means provided by BETA CAE Systems S.A. towards the development of robust, repeatable and realistic crash and safety simulation models and the evaluation of the analysis results. ANSA Data Management facilitates the "replace-include-exclude" procedures that constantly arise during the model build-up, while ANSA Task Manager guides the generation of the simulation models, identifying the dependencies among modeling actions. The simulation model set-up becomes a repeatable and user-independent procedure, safeguarding the model quality and fidelity.

TECHNICAL PAPER -

1. INTRODUCTION

In the framework of virtual prototype development, a great number of discipline models must be built and analyzed for the validation of a new vehicle model design. In the presently CAE-driven design process, the increasing number of vehicle model variants further increases the number of the load-cases that must be studied. With the aid of powerful and "intelligent" batch meshing capabilities, the bottleneck of the CAE turnaround cycle tends to move from the model discretization to the actual discipline model build-up.

The build-up of crash-test simulation models is a process complex and time-consuming, vulnerable to model parameter changes and error-prone. Furthermore, despite of the very detailed specifications provided by OEMs for the set-up of simulation models, the overall model quality is always user-dependant. Thus, the "weak" points in the current crash-test simulation models set-up process can be roughly outlined as follows:

- Organization: there is a great difficulty in gathering all necessary data
- Flexibility: in most cases, CAE engineers have a hard-time incorporating component updates at the late stages of the model build-up
- Repeatability: the simulation model build-up process requires great expertise and cannot be reproduce by inexperienced engineers
- Robustness: small changes in the model parameters may lead to a great scatter in the simulation results
- Model validation: there are incomplete model checks prior to the output

All these deficiencies maintain the CAE productivity at low level.
In this paper, the capabilities of ANSA Task Manager in combination with ANSA Data Management are explored, having as target the elimination of the physical drawbacks that arise during the crash-test simulation models build-up. ANSA Task Manager is used to capture the best practices for the crash simulation models set-up, as these are determined by the CAE analysis expert, scoping to the increase of CAE teams flexibility, efficiency and productivity.
The whole procedure is demonstrated using the full vehicle model shown in Fig.1.

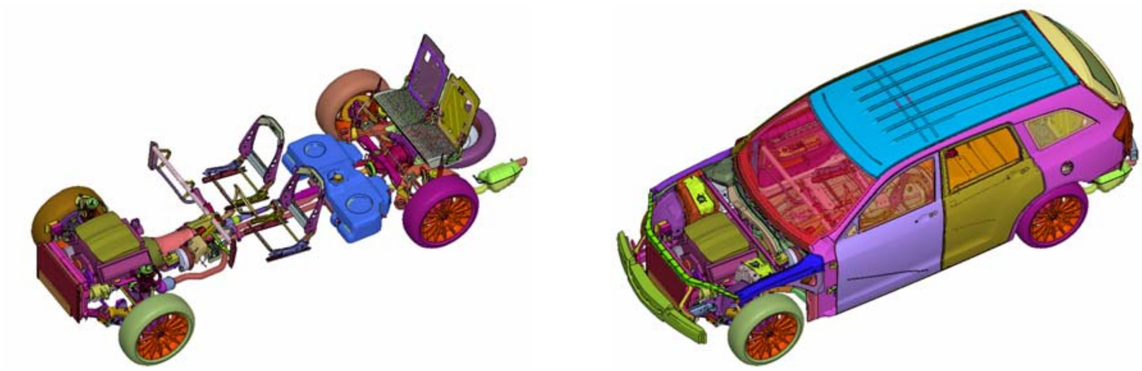


Fig.1. Full vehicle model

From this model, two load-cases input deck files are extracted. The first is a front-impact 40% offset analysis, with an initial velocity of 64km/h (FMVSS 208), to be conducted with LS-DYNA solver, while the second is a side-impact analysis, with an initial velocity of the barrier of 50km/h (EURO-NCAP 2004), to be conducted with PAM-CRASH. The use of two different explicit codes for the analysis of two crash-test simulation models is for demonstration purposes only.

2. ANSA DATA MANAGEMENT AS A DATA POOL

ANSA DM collects and stores in a structured and hierarchical form all engineering data that are necessary during the development process of a vehicle simulation model (1) . Tasks created in ANSA Task Manager are saved in ANSA DM as template processes in order to guide the model build-up of model assemblies. These processes are built once by the CAE analysis expert, who must take into account and include in the tasks all distinct modeling actions that need to be performed during the simulation model build-up.

The great variety of data that will be incorporated in the analysis models studied are gathered under ANSA DM. Such data are:

- Mesh representations of components
- Material databases
- Barrier models
- Custom-made templates for Connector Entities, Mass Trim items, Boundary Conditions and Output Requests Generic Entities

ANSA Task Manager is perfectly combined with ANSA DM in all cases where data storage or data retrieval is necessary. Thus, all templates and auxiliary components referenced by the process items can be automatically invoked and incorporated in the model according to the rules specified in the Tasks.

3. CONCEPTUAL ORGANIZATION OF THE CRASH-TEST SIMULATION MODEL BUILD-UP IN ANSA TASK-MANAGER

ANSA Task Manager considers the model build-up consisting of three distinct stages: The *Common Model*, the *Solver Common Model* and the *Solver Load-case*. Each stage groups several *task items* which represent certain modeling actions. During the *Task* execution, the actions implied by the *task items* are realized and verified. Possible dependencies among modeling actions are automatically detected and treated appropriately.

The *Common Model* contains all modeling actions which are common for all the discipline analyses that will be conducted on the model (i.e. NVH analyses, front impact, side impact, rear impact analyses, durability analyses etc.). Thus, the *Common Model* is stripped of any solution dependant entities and it is ready to be “transformed” in a form suitable for the analysis that follows.

The *Solver Common Model* contains those modeling actions which are not common for all the discipline analyses, but are certainly common for all the load-cases that will follow. The *Solver Common Model* also plays another significant role: It dictates the “transformation” of the *Common Model* into a form suitable for the analysis that will follow.

Finally, the *Solver Load-case* contains the modeling actions that are load-case dependent.

For the build-up of a front-impact and a side-impact analysis, regardless of the solver where the analysis will be conducted, the *Common Model* needs to adopt two different mesh representations. This automatically implies that there must be two *Solver Common Model* groups created, one for the front- and one for the side-impact load-case. Under the *Solver Common Model* items, there may be one or more load-cases, varying on the initial conditions, boundary conditions and output requests e.g. initial velocity and barrier position.

4. THE COMMON MODEL IN ANSA TASK-MANAGER

The *Common Model* defines the model that will participate in the analysis. It consists of all the components, BiW connections, connectors and mass trim items that are common to all disciplines (Fig.2).

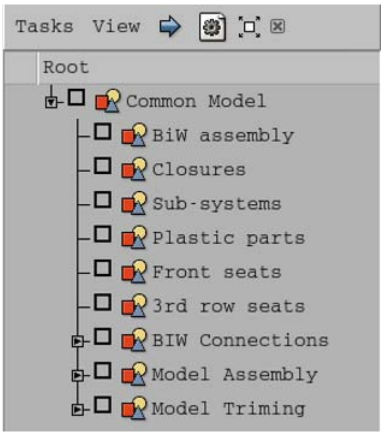


Fig.2. The Common Model task item

The *Common Model*, in the same manner as all other ANSA Task Manager groups, represents a process that must be followed exactly as dictated by the sequence of *Task Items*. This sequence, once defined is saved as a template process in ANSA Data Management system and can be directly retrieved for its application on the assembly model.

<u>Gathering the components that comprise the model</u>

ANSA Task Manager allows for a model grouping suitable for the discipline models build-up. This grouping simplifies the assembly hierarchy tree coming from VPM systems, enables easy access to the actual sub-assemblies and eases the communication of ANSA Task Manager with ANSA Data Management system.

The model grouping created for the goals of this study consists of six sub-models:

- BiW assembly
- Closures
- Sub-systems
- Plastic parts
- Front seats
- 3rd row seats

These sub-models are reflected in the Parts Manager as ANSA groups. These groups initially appear as empty. By simply *linking* the actual model assembly components in the respective Task Manager sub-model groups, ANSA Task Manager becomes aware of the model that it is placed under its control (Fig.3).

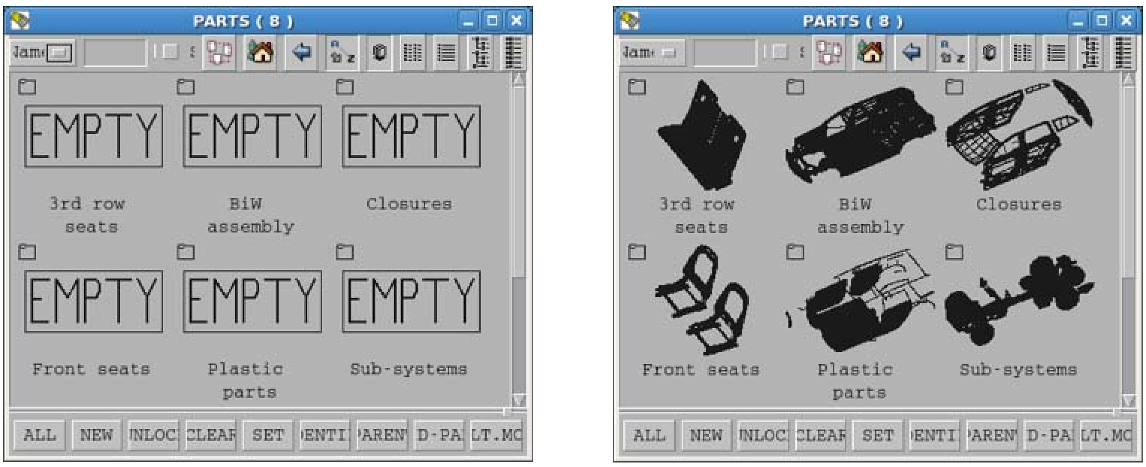


Fig.3. Task Manager sub-models before and after the interfacing with the model assembly

Furthermore, all the capabilities of ANSA Data Management with respect to parts management can be directly accessed from within Task Manager. Most important:
- The retrieval of suitable mesh representations for front- and side-impact analyses
- The notification for part updates and their incorporation in the model assembly.

During the execution of the *Common Model*, ANSA Task Manager verifies that there are no empty parts among the ones referenced by the sub-models. The assembly up to this point is not meshed. The mesh is a discipline-dependent feature.

<u>BiW Connections</u>

After the validation of the sub-model items, comes the BiW assembly. The BiW is assembled using connection points and curves. The welding information will be input at this point. ANSA Task Manager will verify the connectivity specified in each connection entity, with respect to the availability of the parts and properties referenced.

The welding information neutral file is retrieved from ANSA DM. During the execution of the *BiW Connections* group of items and with the aid of the Connection Manager, Task Manger validates every single connection entity (Spot weld points, Spot weld lines, Adhesive lines, Bolts).

The connection entities do not take an FE-representation at this stage. The FE-representation is a discipline-dependent feature.

<u>Model Assembly: The Connector Entities</u>

In order to impose the kinematic constraints that physically exist between parts and sub-assemblies Connector Entities are used (Fig.4). A Connector Entity can carry information regarding:
- Its location in space and orientation.
- Connectivity. Which components are connected?
- Representation. What FE-representation should be used?
- Interface. Which is the interface entity between the representation entities and the connected components?

These entities can apply rigid interface between the connected components or even use template, parameterized entities to model the kinematic constraints. During the execution of the *Model Assembly* group of items, Task Manager validates every single Connector Entity with respect to the availability of the parts and properties referenced in its connectivity information.

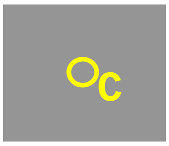


Fig.4. Connector Entity Symbol

At this stage, the Connector Entities do not take any FE-representation. The FE-representation is a discipline-dependent feature.

Model Trimming: The Mass Trim items

Mass trimming takes place either with the addition of lumped mass on certain components or with the addition of distributed mass on components. All cases where mass addition is common for all discipline models are collected under the *Mass Trimming* group in the form of *Mass Trim* items (Fig.5). A Trim Item can carry information regarding:
- Its location in space.
- Connectivity. On which components should the mass be added?
- Representation. What FE-representation should be used?
- Interface. Which is the interface entity between the representation entities and the components?

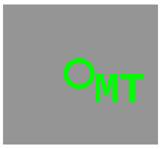


Fig.5. Mass Trim item Symbol

During the execution of the *Model Trimming* group of items, Task Manager validates every single Mass Trim item with respect to the availability of the parts and properties referenced in its connectivity information.

At this stage, the Mass Trim items do not take any FE-representation. The FE-representation is a discipline-dependent feature.

The characteristic features of the model assembly after the validation of all *Common Model* contents are summarized below:

- There are no parts missing among the sub-models that comprise the model assembly
- The welding information exist in the form of connection entities (connection points and connection curves) which verified with respect to their connectivity information
- The locations where kinematic constraints exist between parts and sub-assemblies are marked with Connector Entities. These entities are verified with respect to their connectivity information.
- The locations where mass trimming takes place for all discipline models are marked with Mass Trim items. These entities are verified with respect to their connectivity information.
- There are no intersections between components. All possible intersections where resolved prior to the population of the model geometry into ANSA Data Management[1].

This error free assembly model (Fig.6) comprises the base for the creation of any discipline model, since it is ready to be "transformed" into any suitable form.

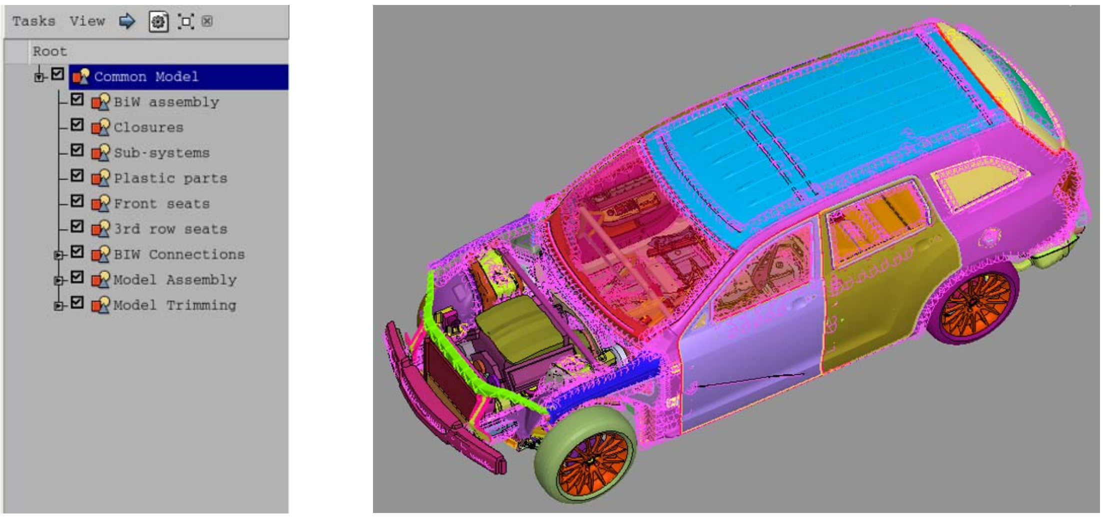


Fig.6. Common Model completed

5. THE SOLVER COMMON MODEL

The significance of the *Solver Common Model* group of items lies in the fact that it is the linkage between the *Common Model* and the *Solver Load-case*. It is the *Solver Common Model* that dictates the "transformation" of the *Common Model* in a form suitable for the analysis to be conducted.

The *Solver Common Model* safeguards a part of the model build-up process in the form of a sequence of modeling actions. Once built by the CAE analysis expert, it is saved as a process template in ANSA Data Management system so that it can be easily retrieved and re-used for several applications.

In this study, an *LS-Dyna Common Model* is used for the build-up of the front-impact load-case and a *Pam-Crash Common Model* for the side-impact one.

<u>LS-Dyna Common Model</u>

As soon as the *LS-Dyna Common Model* is invoked, ANSA Task Manager is notified that the actual model built within the *Common Model* must adopt a form suitable for the LS-Dyna analysis. All the *Common Model* task items that will be affected by the "transformation" of the *Common Model* become automatically un-checked, (Fig.7). Thus, the user is forced to execute them again, this time with the conditions dictated by the *LS-Dyna Common Model*.

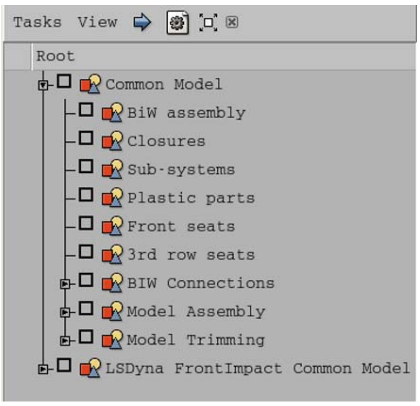


Fig.7. Addition of the LS-Dyna Front-impact Common Model

During the re-execution of the *Common Model*, the components referenced by the sub-model items must adopt a suitable for front-impact analysis mesh representation, the BiW connections must be "realized", connecting the parts they reference by suitable for the LS-DYNA analysis entities, the Connector Entities and the Mass Trim items must be also "realized", using built-in or custom template representations.

Sub-model items: FE-representation

With the *LS-Dyna Common Model* present, ANSA Task Manger does not validate any sub-model item unless the components it references do not contain any unmeshed macro areas or volumes. Therefore, for each sub-model the proper representation is retrieved from ANSA DM, with the aid of the *Part Representation Manager* (Fig.8). All functionality related to the communication of the Task Manager model with ANSA DM is directly accessed from within Task Manager.

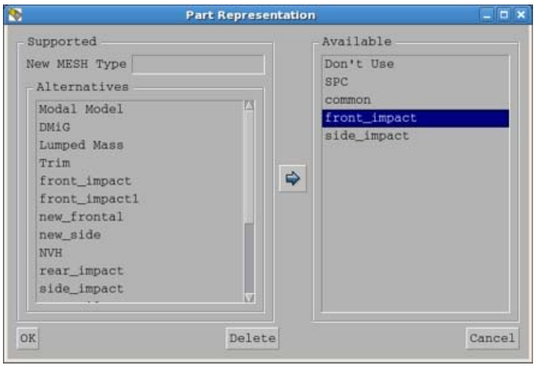


Fig.8. Retrieving the front-impact mesh representation from ANSA DM

After the retrieval of the front-impact mesh representation for each sub-model item (Closures, Sub-systems, Plastic parts, Front seats, 3rd row seats), ANSA Task Manager validates their definition with respect to the existence of unmeshed areas and volumes (Fig.9).

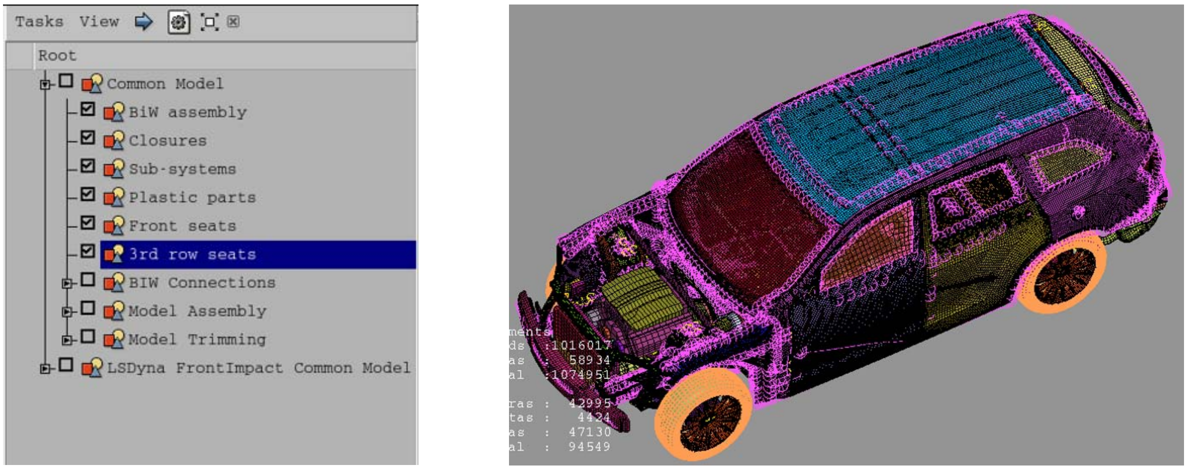


Fig.9. Front-impact mesh representation for *Common Model* sub-models

BiW Connections: Realization of welding information

With the *LS-Dyna Common Model* present, ANSA Task Manger does not validate any BiW connection entity type item unless all the entities of this type are “realized” with suitable FE-representations. For each connection entity type, annotations created by the CAE analysis expert guide the realization pattern (Fig.10).

```
AdhesiveLines:
TYPE: HEXA CONTACT
PSOLID id: 14000000
Search distance: 20
"Fix Hexa Quality": on
```

```
Bolts:
Head type: RBE2
Body type: RBE2
"Attach to": "Hole", "shell"
search distance: 35
```

Fig.10. Annotations for the realization of Adhesive Lines and Bolts

Table 1 shows examples of connection entities realization patterns.

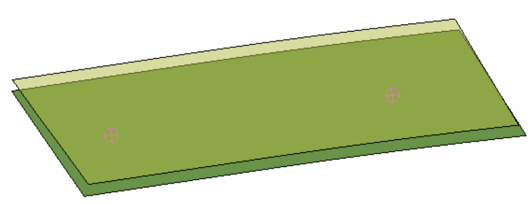
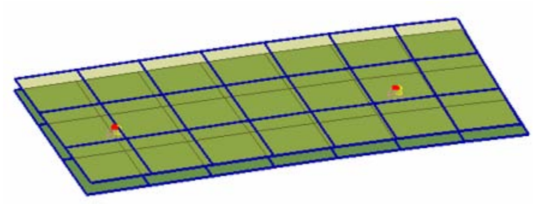
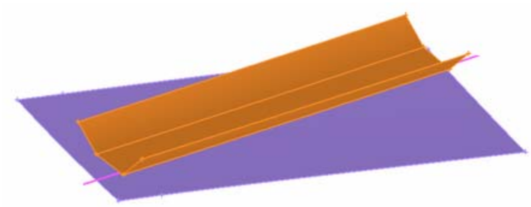
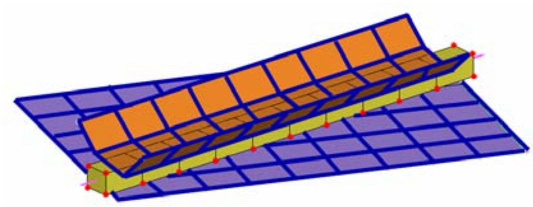
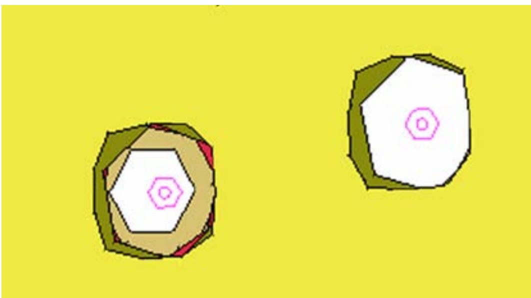
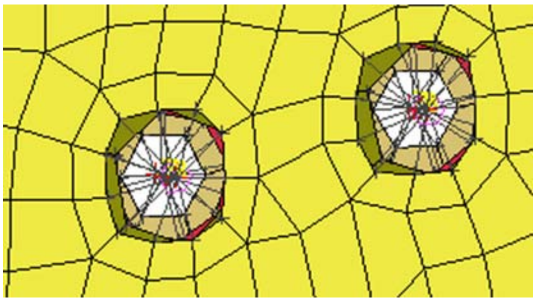
Before the LS-Dyna Common Model	After the LS-Dyna Common Model
 Spot weld point connection entities	 DYNA SPOTWELD pattern
 Adhesive line connection entity	 HEXA CONTACT pattern
 Bolt connection entity	 Rigid interface, rigid body pattern

Table 1. Realization patterns of BiW connections

After the realization of all connection entities, ANSA Task Manager validates their definition.

Model Assembly: Realization of Connector Entities

With the *LS-Dyna Common Model* present during the execution of the *Model Assembly* group of items, ANSA Task Manger applies the Connector Entities FE-representation. Some characteristic features of the Connector Entities realization are:

- The Connector Entities are defined using *mesh density independent* patterns for the identification of connected entities. Thus, a connector can be applied on a hole, a circular feature, a certain number of nodes or elements in a pre-defined search domain
- Connector Entities that connect a rigid with a deformable component with rigid interface automatically detect the case and create *CONSTRAINED_EXTRA_NODEs instead of *CONSTRAINED_NODAL_RIGID_BODY entities
- Connector Entities that connect rigid components together with rigid interface, automatically detect the case and create *CONSTRAINED_RIGID_BODY entities
- The Connector Entities can use the built-in representations (i.e revolute and spherical joints) or even custom made ones, retrieved as templates from ANSA DM libraries, promoting the model robustness

Table 2 summarizes the custom templates used for the representation of Connector Entities. Such templates once created, are saved in ANSA DM libraries so that they can be re-used for various applications.

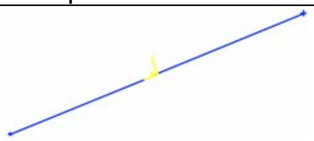
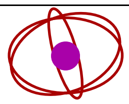
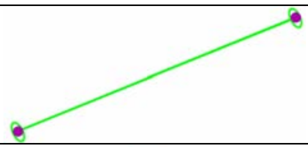
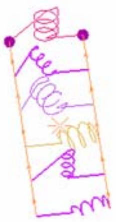
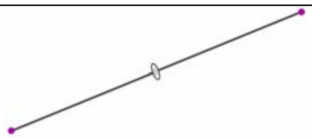
Template in ANSA DM	Description
	*ELEMENT_BEAM_ELFORM_1 Parameters: diameter, length and orientation according to application
	*CONSTRAINED_JOINT_SPHERICAL Parameters: Value of added mass on both ends
	*CONSTRAINED_JOINT_REVOLUTE Parameters: Value of added mass on both ends, orientation according to application
	*ELEMENT_DISCRETE for 6 dofs Predefined: *DEFINE_SD_ORIENTATION vectors, *SECTION_DESCRETE Parameters: Value of added mass on both ends
	*CONSTRAINED_JOINT_UNIVERSAL Parameters: Value of added mass on central nodes, orientation according to application
	*CONSTRAINED_JOINT_CYLINDRICAL Parameters: Value of added mass on both ends, orientation according to application

Table 2. Parameterized templates for connectors retrieved from the ANSA DM library

Figure 11 shows an example of connector entities realization.

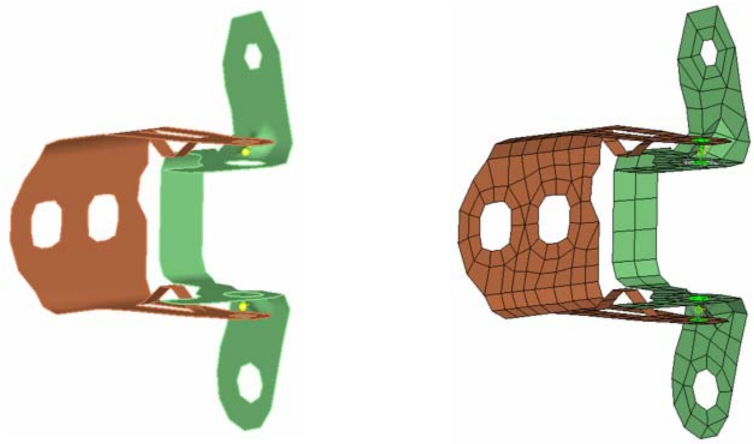


Fig.11. Connector entities before and after their realization

Model Trimming: Realization of Mass Trim items

With the *LS-Dyna Common Model* present during the execution of the *Mass Trimming* group of items, ANSA Task Manger applies the Mass Trim items FE-representation. The FE-representations of Mass Trim items for LS-Dyna can be:

- Certain amount of added mass distributed over pre-defined components in the form of *ELEMENT_MASS
- Certain amount of added mass attached on pre-defined components at specified locations in the form of *lumped mass*
- Certain amount of added mass distributed over pre-defined components in the form of a skin of *MAT9_MAT_NULL. The density of the material is automatically calculated so as to result to the desired total weight
- The substitution of the detailed FE-representation of a component by an equivalent amount of mass distributed over the nearby components

After the realization of Mass Trim items with the pattern dictated by the *LS-Dyna Common Model* the *Common Model* has adopted a form suitable for the front-impact analysis.

Front-impact related items

The items added by the *LS-Dyna Common Model* are common for all the front-impact load-cases that may follow. Such items can be additional components, boundary or initial conditions, output requests and of course model checks. In this study, the first items to be added by the *LS-Dyna Common Model* are the additional components and the change of the detailed representation of certain components into a reduced one, suitable for the front-impact analysis:

- Passengers' mass: These mass elements of pre-defined value are added in the form of *lumped masses* and are attached specified components (i.e. seats, rear floor) with the aid of Mass Trim items.
- Instrumentation mass: This mass of pre-defined value is added in the trunk in the form of *lumped mass* and is attached to specified components, again using the Mass Trim items.
- Closures interior plastic components: The detailed FE-representation of these components is substituted by a *Trim representation*, spreading mass elements of an equivalent total weight over the nearby inner panel components (Fig.13)

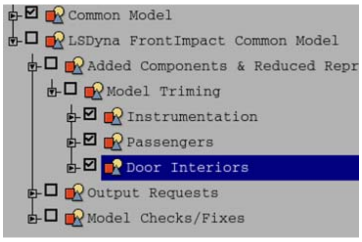


Fig.12. Front Impact added components

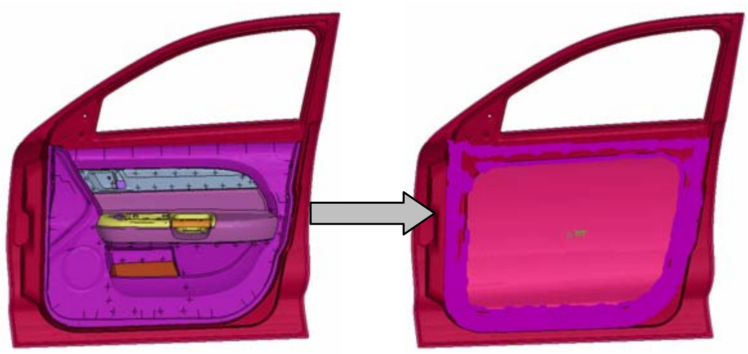


Fig.13. Trim representation created with the aid of Mass Trim items

The creation of section forces output requests, a tedious and time-consuming procedure, becomes really flexible and efficient with the use of Output Request Generic Entities. The positions, cutting planes and the components cut are pre-defined in the Output Request Generic Entities by the CAE analysis expert. Once defined, they can be applied during the Task execution with no extra input.

Output Request Generic Entities are also used for the positioning of accelerometer sensors at pre-defined locations. The representation of accelerometer sensors, again pre-defined by the CAE analysis expert, is stored as custom template in ANSA DM library (Table 3). Each output request definition contains information for the location and the connectivity of the accelerometer sensor to be created. Figure 14 shows the accelerometers and sections created by Output Request Generic Entities.

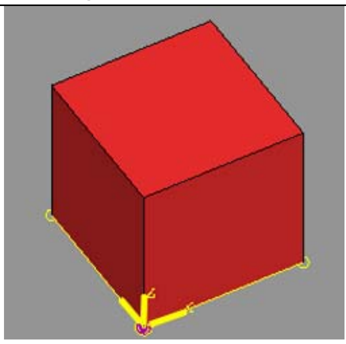
Template in ANSA DM	Description
	*ELEMENT_SEATBELT_ACCELEROMETER Each accelerometer is attached on a 10x10x10 rigid hexa element. A mass element is attached to the accelerometer reference node.

Table 3. Custom template for accelerometer sensor retrieved from the ANSA DM library

Fig.14. Accelerometers and sections created by Generic Entities

After the addition of output requests, the model integrity is checked. Among the various available checks, the CAE analysis expert is flexible to add custom checks with the aid of user scripts. ANSA Task Manager does not proceed until all checks are successfully implemented. The checks performed at this stage for this study are summarized in Table 4.

Model check	Fix options
Rigid dependency	After the identification of rigid dependency errors, they can be automatically fixed from within ANSA Task Manager
Connectivity	Loose components are directly identified
Joints	The validity of joints definition is directly checked
Number of integration points	This custom made check assigns 5 IPs in case there are properties that reference less
Undefined materials	Material properties are automatically updated from the LS-Dyna material database retrieved from ANSA DM

Table 4. Model integrity checks in *LS-Dyna Common Model*

After the implementation of model checks and the correction of possible errors the *LS-Dyna Common Model* is ready (Fig.15). The model built up to this stage can be used for the creation of any LS-Dyna front-impact load-case.

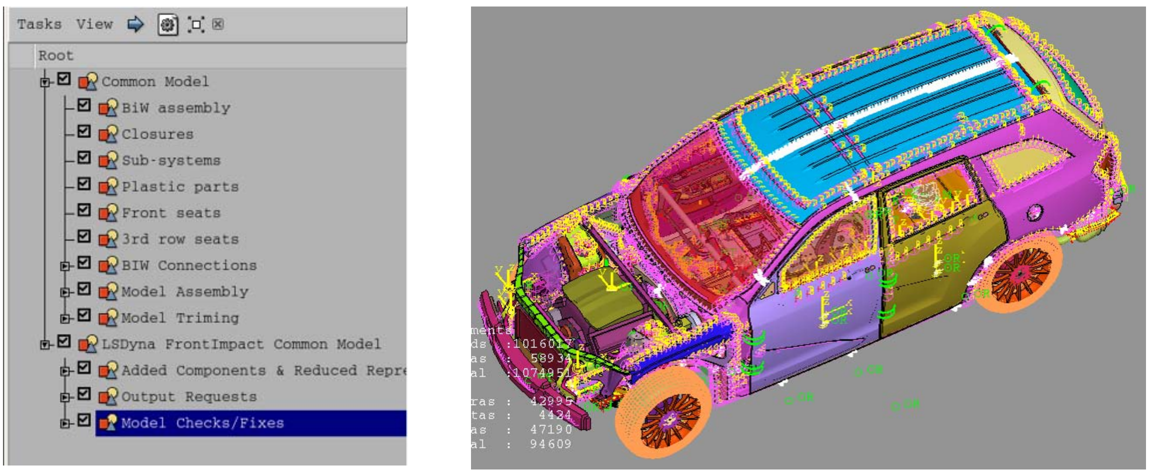


Fig.15. LS-Dyna front-impact common model completed

PAM-CRASH Common Model

The *Pam-Crash Common Model* for the side-impact load-cases is very much alike the *LS-Dyna Common Model* with respect to the work-flow. At a first step, the *Pam-Crash Common Model* “transforms” the *Common Model* in a form suitable for side-impact analysis. Then, it adds to the model certain solver dependant entities that are common for all the side-impact load-cases that may follow.
As soon as the *Pam-Crash Common Model* is invoked, the *Common Model* gets un-updated so that the user executes it again, with the conditions dictated by the *Solver Common Model*.

“Transformation” of the Common Model

With the aid of the Part Representation Manager, the *Common Model* sub-models adopt a mesh representation suitable for side-impact analysis, as shown in figure 16. The mesh representation of each component is invoked from ANSA DM.

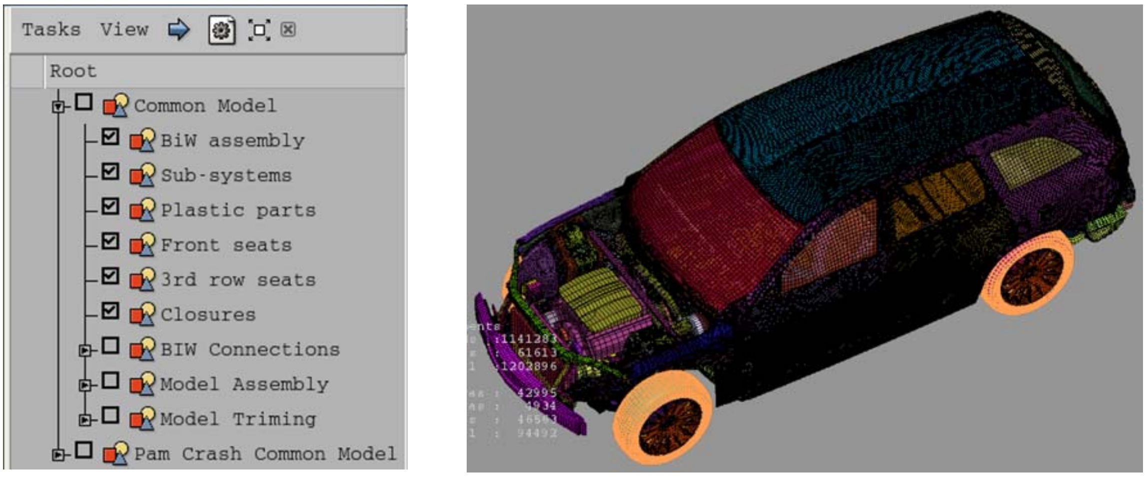
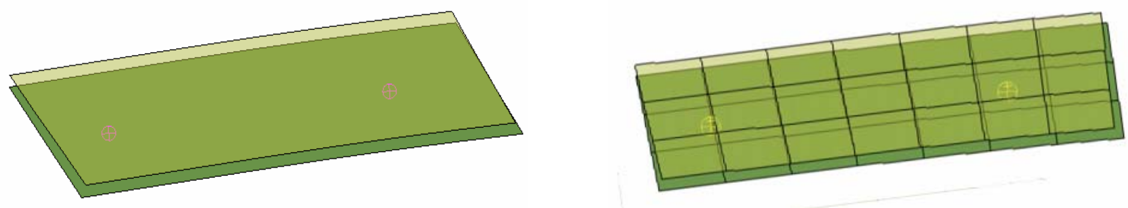


Figure16. Side-impact mesh representation for Common Model sub-models

The BiW Connections are “realized”, getting a suitable FE-representation. Again, the user is guided by annotations for the attributes of the connection elements that will be created by the Connection Manager. The spot weld points and curves are realized as PLINK elements (Fig.17), the adhesive lines as HEXA-CONTACT and finally the bolts with rigid interface and rigid body.



Spot weld point connection entities PAM PLINK pattern

Fig.17. Realization pattern of BiW connections: Spot weld points

For the Connector Entities, which take care of the kinematic constraints between components, proper templates are retrieved from the ANSA DM libraries. KJOIN, BEAM and SPRING elements with predefined properties are used this time.

Finally, the Mass Trim items are realized as ADMAS entities, either distributed over components or as lumped masses with RBODY interface.

Side-impact related items

The items added by the *Pam-Crash Common Model* are common for all the side-impact load-cases that may follow. Starting from the additional components, again the passenger's mass and the instrumentation mass are added with the aid of *Mass Trim* items. The closure inner plastic components are also turned into the reduced *Trim* representation.

Section forces output requests are created with the aid of pre-defined Output Request Generic Entities.

Finally, the integrity of the model is checked with the aid of built-in and custom checks. The CAE expert who built the task has pre-defined the checks that must be performed prior to the load-case dependant entities addition (Table 5).

Model check	Fix options
Rigid dependency	After the identification of rigid dependency errors, they can be automatically fixed from within ANSA Task Manager
Connectivity	Loose components are directly identified
PLINKs	The validity of PLINK elements definition is directly checked
Number of integration points	This custom made check assigns 5 IPs in case there are properties that reference less
Undefined materials	Material properties are automatically updated from the Pam-Crash material database retrieved from ANSA DM

Table 5. Model integrity checks in *Pam-Crash Common Model*

After the implementation of model checks and the correction of possible errors the *Pam-Crash Common Model* is ready (Fig.18). The model built up to this point can be used for the creation of any Pam-Crash side-impact load-case.

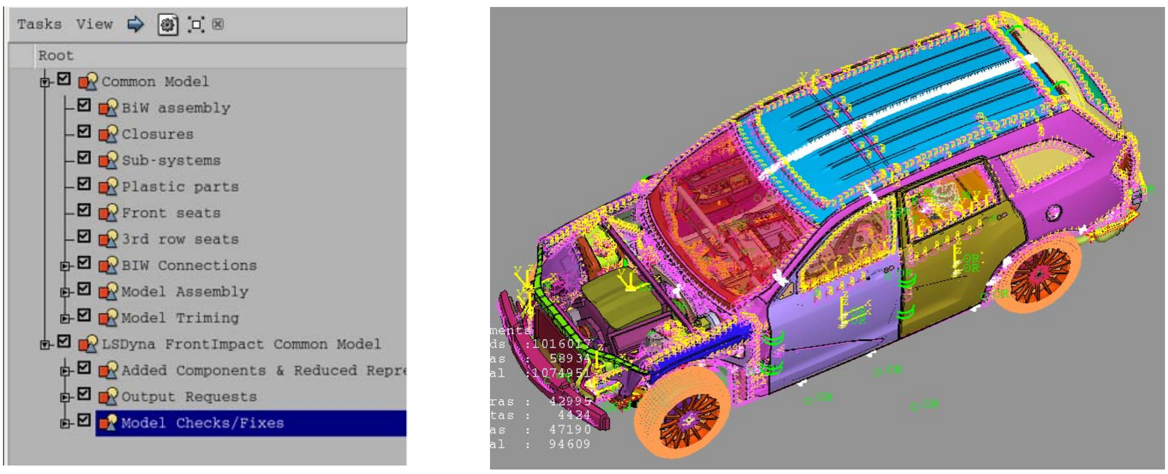


Fig.18. Pam-Crash side impact common model completed

6. THE SOLVER LOAD-CASE

All the entities that differentiate the front-impact load-cases from each other are added by the *Solver Load-case* group of items. This Task is again invoked from the ANSA DM data pool. The most important aspects of the *Solver Load-case* Task are outlined below:

- Solver controls: The solver controls are set-up once for each load-case by the CAE analysis expert and are safeguarded in the Task
- Contact interface cards: The contact interface cards, along with their parameters, are set-up once during the Task build-up. During the Task execution, Task Manager automatically fills the contact sets and there is no need for user-intervention in the contact definition.
- Barrier file definition and positioning: The barrier file to be used in each load-case is pre-defined in the *Barrier Positioning* item of ANSA Task Manager along with information for the positioning procedure. During the Task execution, the barrier is retrieved from the ANSA DM data pool and is automatically positioned and de-penetrated according to the load-case specifications.
- Rigid road: The attributes of the rigid wall used as the boundary road are saved along with the Task.
- Initial velocity and acceleration field: The solver cards parameters are defined once by the CAE analysis expert. There is no need for user intervention during the Task execution, since Task Manager automatically fills the sets referenced in these cards with the appropriate entities
- Model checks: The model is checked prior to the output with respect to load-case specific entities definition.

Front-impact 40% offset 64km/h with LS-DYNA

All the aforementioned *Solver Load-case* attributes find application in the front-impact load-case. Additionally, in this *LS-Dyna Load-case* nodal time history is requested at pre-defined locations with the aid of Output Request Generic Entities. The custom representation of these output requests is retrieved from the ANSA DM library (Table 6).

Template in ANSA DM	Description
	*DATABASE_HISTORY_NODE_SET

Table 6. Custom template for nodal time history retrieved from the ANSA DM library

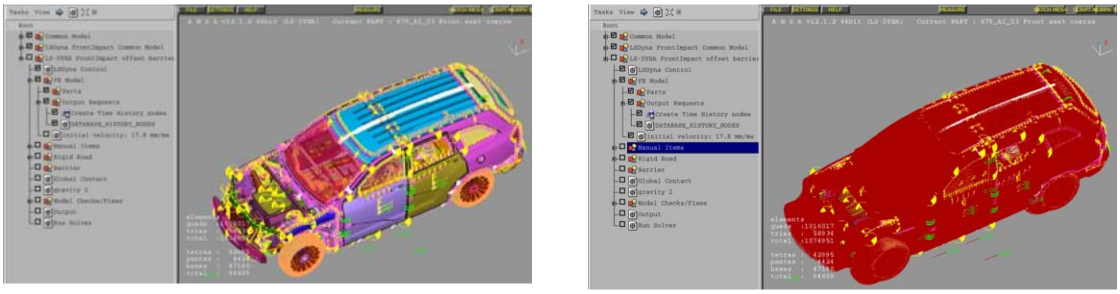


Fig.19. Addition of time history nodes and initial velocity

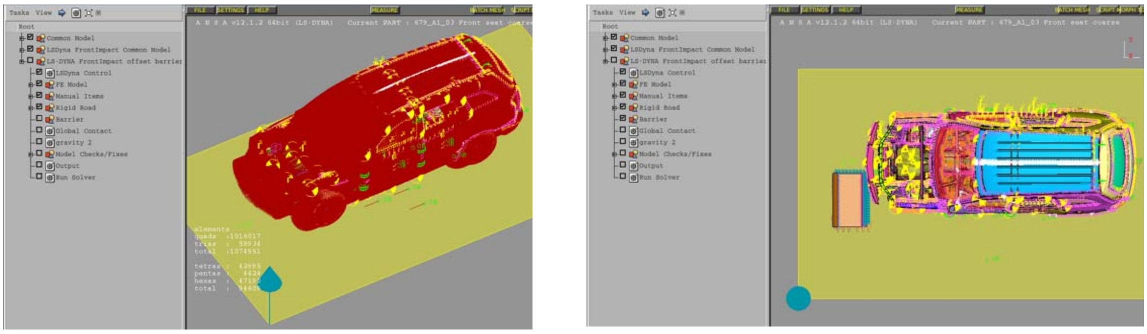


Fig.20. Addition of rigid-road and front barrier

Figures 19-20 show the items added sequentially by the *LS-Dyna Load-case*.

The *Barrier Positioning* tool assures that the barrier will be positioned properly relatively to the vehicle body, with no penetration between them. Figure 21 shows the *Barrier Positioning* control card, which requires the minimum information for a front-impact load-case.

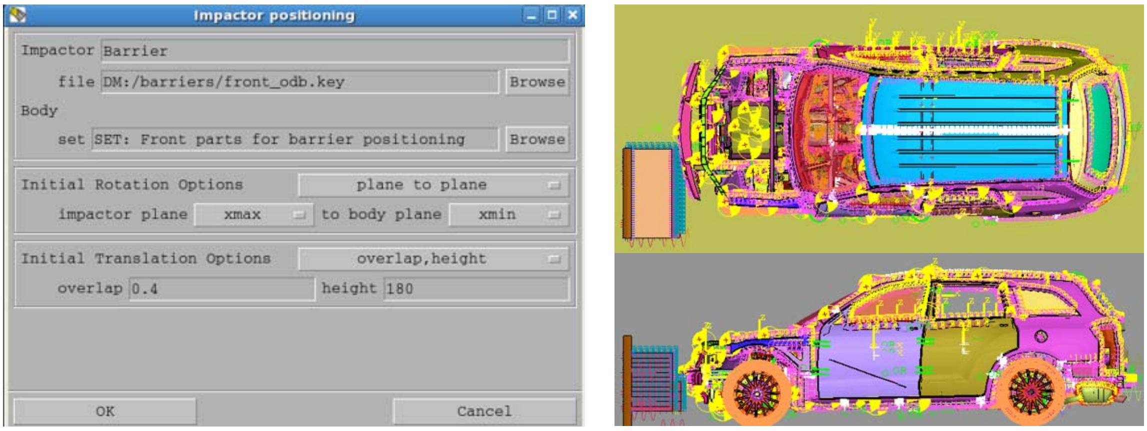


Fig.21. Barrier positioning: Pre-defined set-up of offset and height

Finally, the global contact definition and gravity are added and the model is checked with respect to load-case specific definitions. After the execution of model checks, the analysis model is ready for output. The complete model is shown in figure 22.

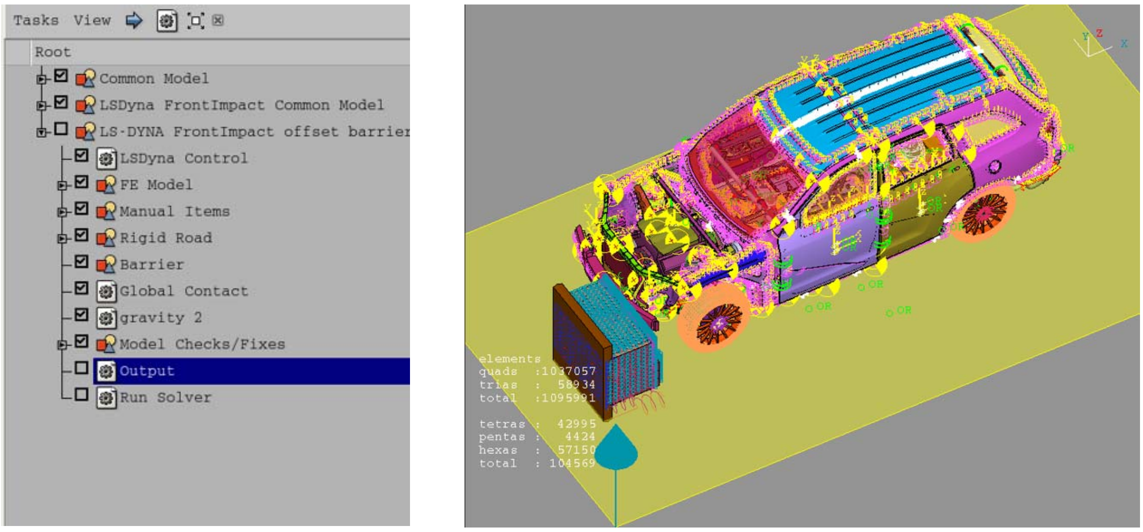


Fig.22. LS-Dyna Front-impact 40% offset 64km/h complete model

Side-impact 50km/h with Pam-Crash

Load-case specific entities are added in the *Pam-Crash Load-case*, in the same manner as in the front-impact case. The following images show the sequential addition of load-case specific entities.

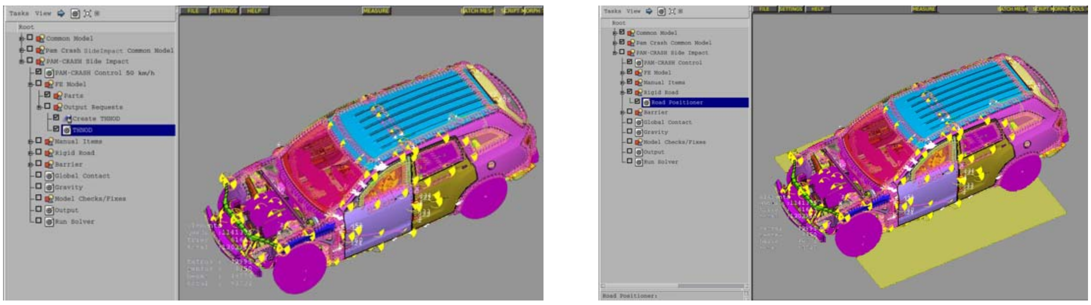


Fig.23. Addition of time history nodes and rigid-road

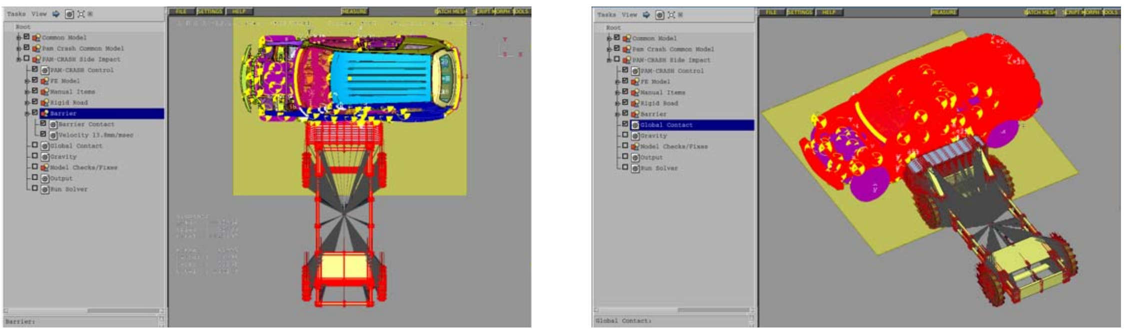
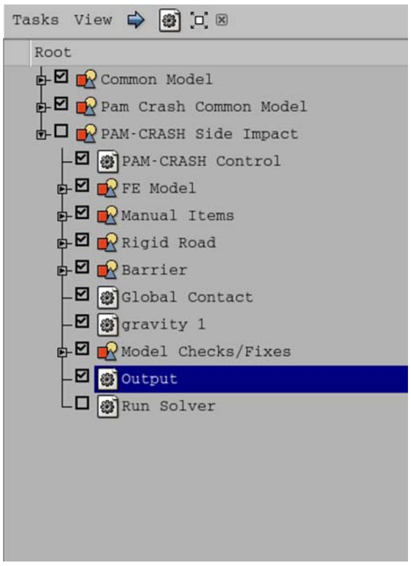


Fig.24. Positioning of the barrier and definition of the vehicle-barrier contact
Addition of vehicle global contact and barrier initial velocity

Finally, the gravity acceleration field is added and the model is checked with respect to load-case specific definitions. After the execution of model checks, the analysis model is ready for output. The complete model is shown in figure 25.



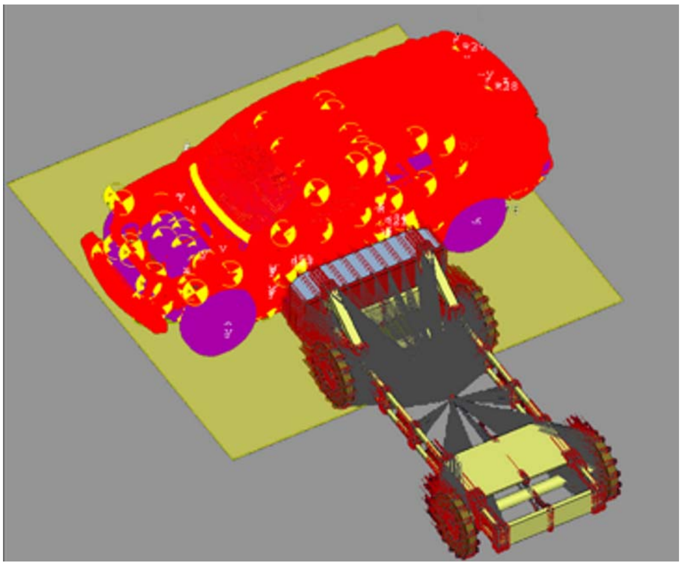


Fig.25. Pam-Crash Side-impact 50km/h complete model

7. CONCLUSIONS AND REMARKS

The data organization and process standardization necessary for the demanding task of crash-test simulation models build-up can be achieved with the use of ANSA Task Manager in combination with ANSA Data Management. With the aid of ANSA Task Manager, OEMs can safeguard the model quality and promote knowledge transfer, capturing the best-proven practices for the analysis model build-up as a sequence of modeling actions. ANSA Data Management assures the organization of all data, storing them in a structured form under a common location, enabling their easy retrieval by Task Manager.

Tasks in Task Manager are created by the CAE analysis expert, who collects all the modeling actions and considerations that must be taken into account during the model build-up and interprets them as distinct task items. The completed tasks are saved in ANSA DM as template processes and can be reused for application on the vehicle model assemblies.

ANSA DM libraries carry custom template definitions for connectors, output requests and boundary conditions, assuring that model entities are defined with the proper parameters. Multi-parametric solver cards (e.g. contact definitions, initial velocity, acceleration field, solver controls) are incorporated in ANSA Task Manager with parameter values set by the CAE analysis expert, eliminating error-prone procedures and promoting the model robustness.

During the execution of pre-defined Tasks, ANSA Task Manager makes sure that all task items are properly executed, considering at the same time possible dependencies between them. The validity of model entities definitions is checked prior to the output with the aid of various built-in check algorithms. The model quality is safeguarded and the build-up of crash-test simulation models becomes fast and efficient.

REFERENCES

(1) Georgios Nikolaidis, Dimitrios Zafeiropoulos, Konstantinos Ntamagkas, “Building discipline specific FE-models with the Common Model concept”, 2nd ANSA-μETA International Congress.