DESIGN AND ANALYSIS OF LAUNCHER TANKS WITH BOLTED JOINTS

Jacques Marchesini (1), Marie Lequoy (1), Reinhard Helfrich (2)

(1) INTES France SARL, 22 Rue Sadi Carnot, 78120, Rambouillet, France
EMail: Jacques.Marchesini@intes.fr
(2) INTES GmbH, Schulze-Delitzsch-Str. 15, 70565, Stuttgart, Germany
EMail: Reinhard.Helfrich@intes.de

ABSTRACT

Launcher and satellite tanks for liquid fuels are often almost axisymmetric except of ribs, reinforcements, or bolted joints, for example. The design and modelling of axisymmetric parts or assemblies is much easier and faster, but non-axisymmetric features have to be taken into account in an efficient manner. The innovative concept to achieve that is to combine design and modelling in one single step using one single tool. Instead of doing design with a CAD tool and modelling with a pre-processor, the new concept unifies both steps by defining the geometry and the related Finite Element (FE) mesh together. The advantage of the unification is that the analyst can answer questions about the structural behaviour before a CAD design is available and beyond that, the CAD design can already rely on structural analysis results, which would not be available in the classical sequence of design and modelling.

Fastening components together using bolted joints is very popular mainly due to following reasons. First, the assembly of parts can be organized in a very flexible way. Second, once fastened the joint can be easily released for different reasons like disassembly, exchange of parts, or inspection. Beside the dimensions of bolts (and nuts), the bolted joints are characterized mainly by their pretension force, which is applied by a torque wrench, for example. This pretension force is sticking the parts together including sealing effects.

The presentation will show a recently realized and highly efficient concept for design and modelling of launcher structures with or without enclosed liquid fuels. The generated models can directly be analysed using static or dynamic solvers for interactively improving the design. Pre-stressed bolted joints are easily added to the model, while the bolt pretension itself is applied in a fast contact analysis. The achieved contact status can be used later for subsequent analyses, which may include linear or nonlinear buckling analysis and dynamic analysis including fluid-structure coupled vibrations.

An industrial example is used to demonstrate the capabilities of the new concept. The software for design and modelling is PCGen (a tool in the framework of the VisPER FE pre- and post-processor). PERMAS is used for the various FE analysis steps.

1. THE ENVIRONMENT OF LAUNCHER DEVELOPMENT

The environment of the launcher development is often characterized by the fact that not all data are known at one point in time due to frequent design changes. But developers need a way to study the effect of design changes on the structural behaviour without making a new CAD model. Despite this need, model geometry and corporate standards have to be taken into account even for small and effective studies in order to fulfil all documentation needs for a large project like a launcher development. Fig. 1 sums up the major factors influencing the model creation. In this paper, we will show how this new approach addresses these factors, making the simulation process more flexible and enabling faster design loops.

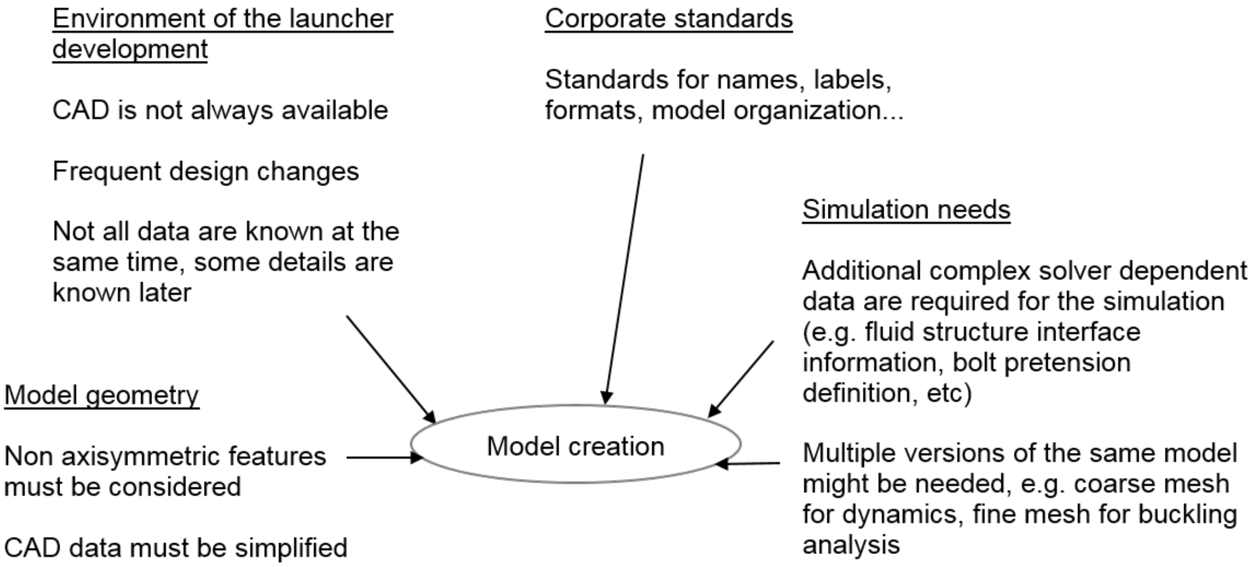


Fig. 1: Factors influencing the model creation

The innovative concept to achieve that is to combine design and modelling in one single step using one single tool. Instead of doing design with a CAD tool and modelling with a pre-processor, the new concept unifies both steps by defining the geometry and the related Finite Element (FE) mesh together (Fig. 2).

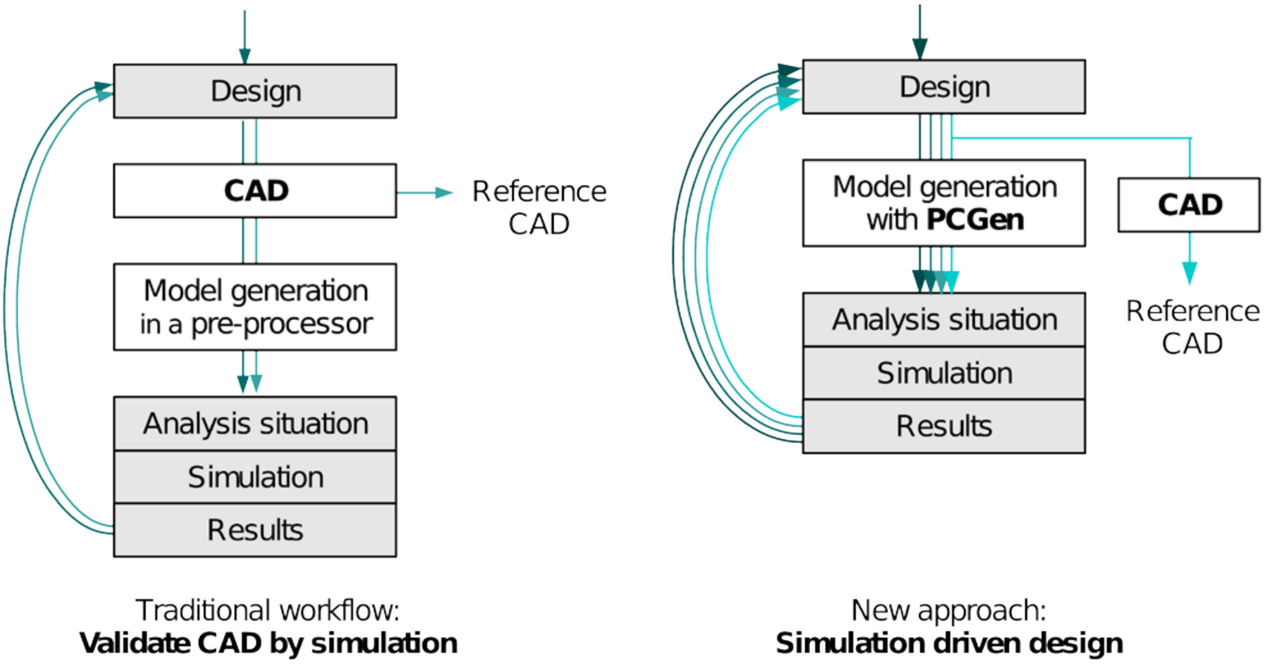


Fig. 2: Comparison between traditional workflow and new workflow

The advantage of the unification is that the analyst can answer questions about the structural behaviour before a CAD design is available and beyond that, the CAD design can already rely on structural analysis results, which would not be available in the classical sequence of design and modelling.

The objective of CAD not the simulation. The CAD step and the subsequent mesh generation in a pre-processor are two complex and expensive steps that require expert software users. Not all CAD data are relevant for the FE simulation, in fact, the meshing process often implies a prior simplification of geometry that represents a step back toward design parameters (e.g. the extraction of the middle planes for shell elements creation).

On the contrary, a global approach enable the software to speak the language of designers and analysts. This is made possible by the use of parameterized structure parts, so that user inputs are kept to the design level (radii, height, thickness, etc.), making the use of the software tool accessible to any designer or analyst. Models also adapt more easily to design change, which makes this approach so helpful in the first design loops of a product. Separate mesh parameters enable the analyst to generate multiple FE models fit for various analyses (e.g. coarser mesh for dynamics, detailed model of bolted joints for contact analysis, etc.), see Fig. 10.

2. QUASI AXISYMMETRIC GEOMETRY

Non-axisymmetric features

Launcher and satellite tanks for liquid fuels are often almost axisymmetric except of ribs, reinforcements, or bolted joints, for example. The design and modelling of axisymmetric parts or assemblies is much easier and faster, but non-axisymmetric features have to be taken into account efficiently.

In PCGen module, models are based on axisymmetric shells, created by the revolution of simple geometrical lines. Additional layers of detail can then be added to create stiffeners, bolted flanges, solid parts, local equipment and other non-axisymmetric features, that can evolve independently from the basic contour. For example, chapter 4 presents some bolted flanges generated with PCGen. Though the geometry is repetitive, each bolt bears complex additional data that makes the model creation complex.

As another example, Fig. 3 shows the resulting displacements and Von Mises stresses in a pressurized tank after a nonlinear buckling analysis. Inner stiffeners were considered. The upper and lower rings, that are thick structures transmitting the efforts from and to adjacent components, were modelled with solid elements, so there stiffness would be more representative. Finally, for a more accurate modelling of the stiffness, only quad shell elements are used for the domes.

Mesh quality

Finally, by opposition to automatic meshes based on an external CAD, the applied meshing knows about the symmetries and therefore generates perfectly symmetric meshes where required, using local cylindrical systems for the coordinates, as well as a mechanism of variables and repetitions allowed by the PERMAS format. Coordinates are generated in double precision. The mesh quality has a direct impact on the solution, for example on the frequency of symmetric modes.

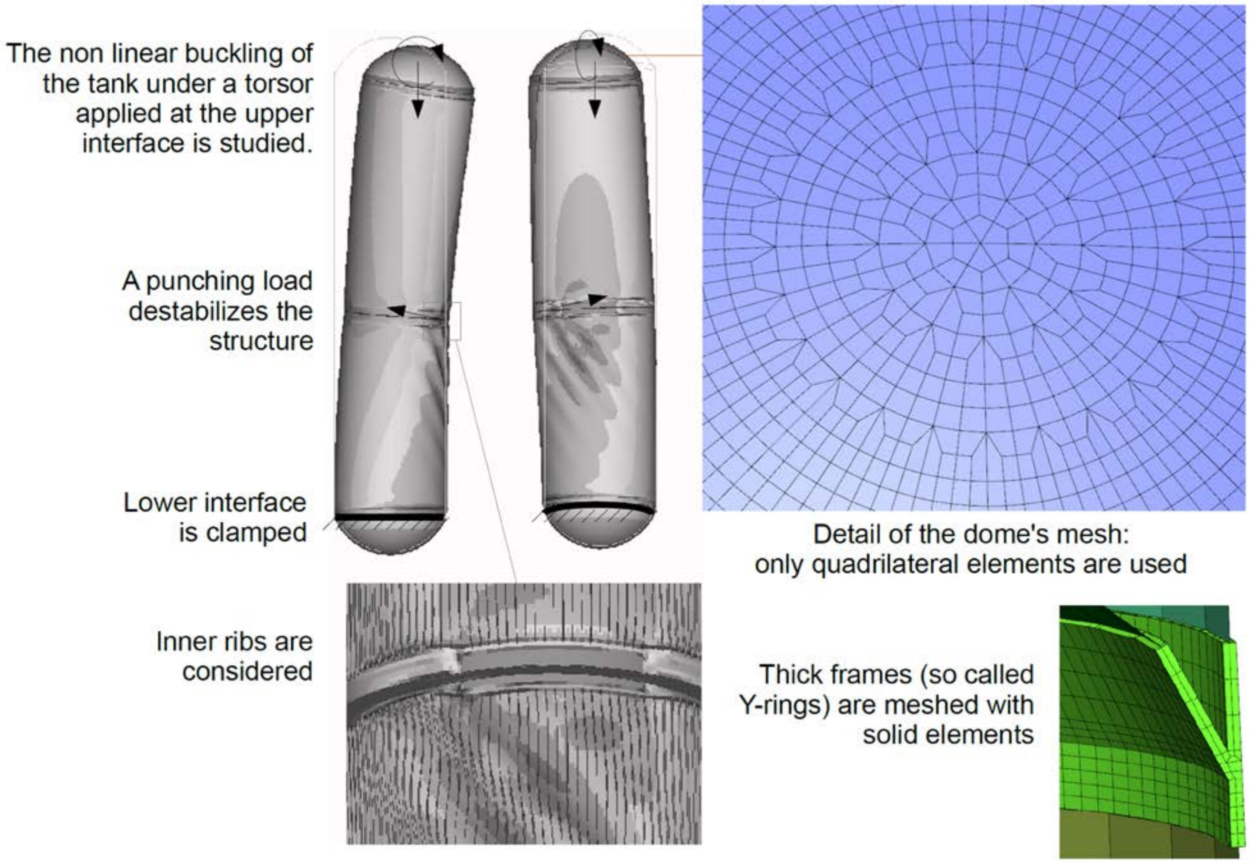


Fig. 3: Buckling analysis of a pressurized tank, view of the model

3. COMPLEX PHYSICS AND VARIOUS ANALYSES

Launcher models contain complex physics such as fluid-structure coupled acoustics or contact. So, their simulation requires much more than a mesh.

The model complexity lies in its physics

In the highly cooperative environment of launcher development, finite elements models, as any other shared piece of data, are constrained in their content and format to corporate standards, defining for example how labels and names are assigned, how models are organized and validated. These standards increase the operator's burden as they are not implemented in general tools, but they also pave the way for automation.

Moreover, most of the cost of model generation in a pre-processor comes from the additional simulation data. Those data are not derived from the CAD, but are defined by corporate standards and know-how, solver requirements, and must be adapted to each study. The specific approach here allows the integration of such standards in the model generation tool, thus relieving the burden of the operator.

Various models are needed for various analysis

Since the approach decouple the mesh parameters from other data, it makes it easy to generate various models for various analyses, for example coarse meshes for dynamics and fine mesh for static and nonlinear buckling. It is also easier to remove details from a model than with a classical workflow.

Model qualification

Finally, the module PCGen has some additional features for model qualification, like the creation of model and command files for traditional model tests, e.g. weight and static pressure validation. PCGen can then compare the results with analytical values, and export correction coefficients. The user can also export tables, reports and blueprints in SVG format.

4. APPLICATION TO BOLTED JOINTS

Fastening components together using bolted joints is very popular mainly due to following reasons. First, the assembly of parts can be organized in a very flexible way. Second, once fastened the joint can be easily released for different reasons like disassembly, exchange of parts, or inspection. Beside the dimensions of bolts (and nuts), the bolted joints are characterized mainly by their pretension force, which is applied by a torque wrench, for example. This pretension force is sticking the parts together including sealing effects.

The cost of modelling and simulation

Bolted flanges are repetitive yet complex structures. Once again, the complexity lies in the simulation data and hypotheses, e.g. how the bolts and flanges connect (see Fig. 4).

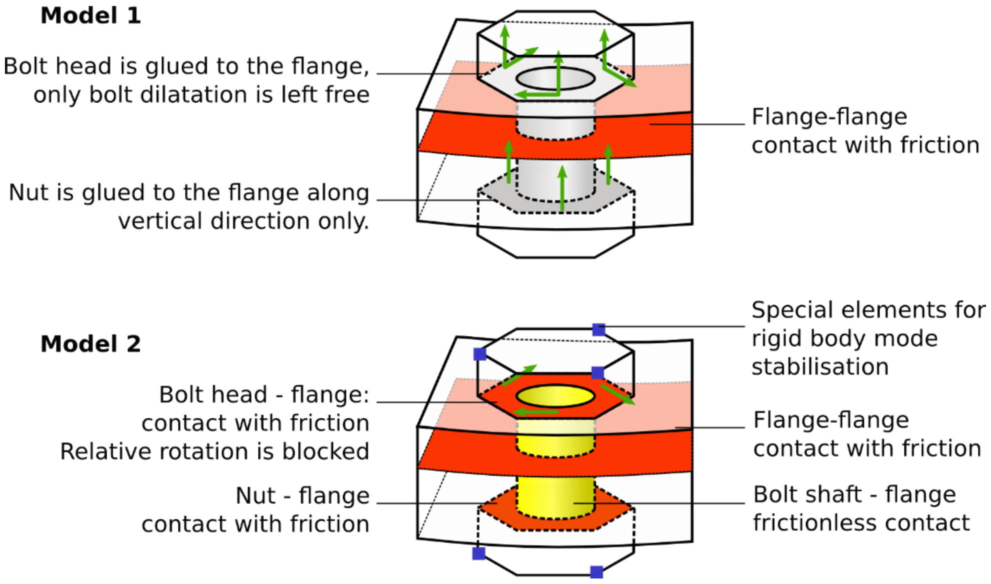


Fig. 4: Two examples of bolt stabilisation models

Fig. 5 shows how high level information that the analyst holds, like bolt class type and the load values, translate into the final FE model. A great deal of this complexity is hidden to the operator.

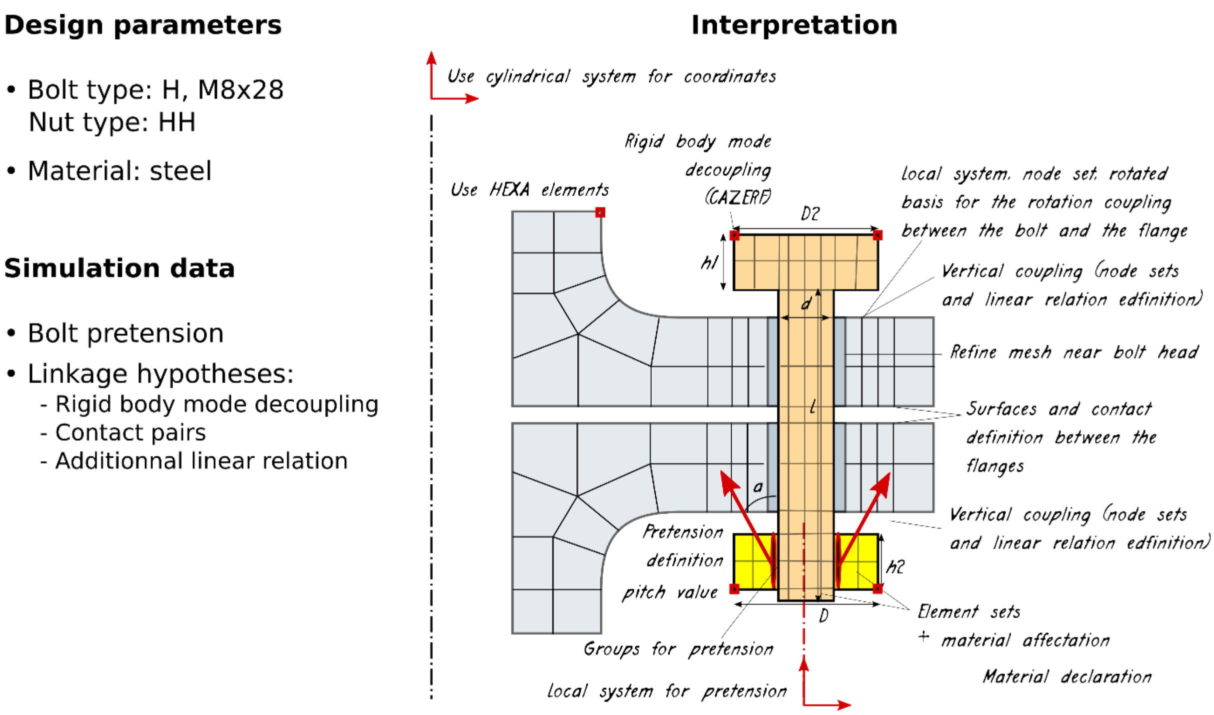
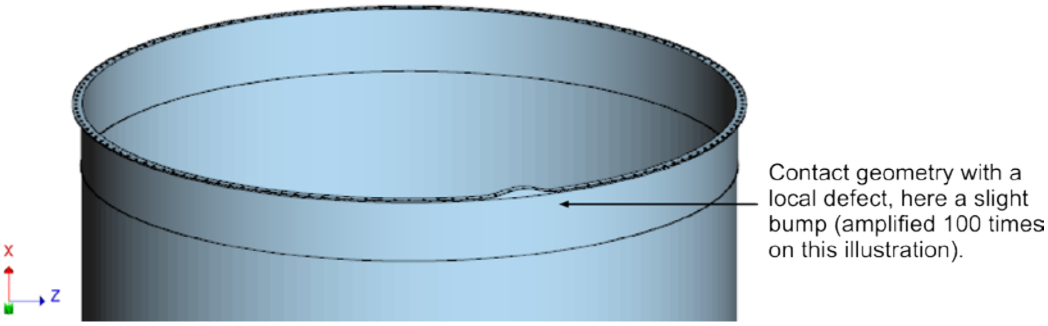


Fig. 5: How information about the design and analysis hypothesis of the bolted flange translates into the final FE model.

Also, the hypotheses taken by the solver can be difficult to set right. PERMAS uses a Coulomb law for friction by default, and considers internally the contact problem as an optimization problem, thus suppressing the need for penalty coefficients. This approach also makes it possible to accelerate the convergence of the solution for cyclic excitations. Moreover, PERMAS provides special zero force springs that greatly ease the rigid body mode stabilization: during the contact iterations, their contribution is nullified and they do not alter the final solution.

Finally, contact analysis is highly CPU expensive, especially when friction is considered. But this cost becomes more and more affordable as hardware and software performance increase. In the following cases, acceleration by using GPU has considerably shorten the run times.

<u>Example 1: Flange geometry and pretension order</u>



Contact geometry with a local defect, here a slight bump (amplified 100 times on this illustration).

Fig. 6: Local bump in the contact geometry

The stress in bolts is also studied to check the reliability of the assembly. What is the effect of the imperfect flange geometry? How does the stress in the bolts depends on the pretension order? To address these questions, a model of the complete bolted joints is required.

The following example shows a simple bolted flange (144 bolts) joining two aluminum cylinders (radius: 1m). A local defect in the contact geometry is introduced as a function of the coordinates in a cylindrical system, see Fig. 6. The model was generated with PCGen and VisPER. The lower interface is clamped, the bolt pretension force is 4000 N. The friction is always considered and the sequence of the bolt pretension is investigated. We show here the final traction load in each bolt of the joint setup (see Fig. 7).

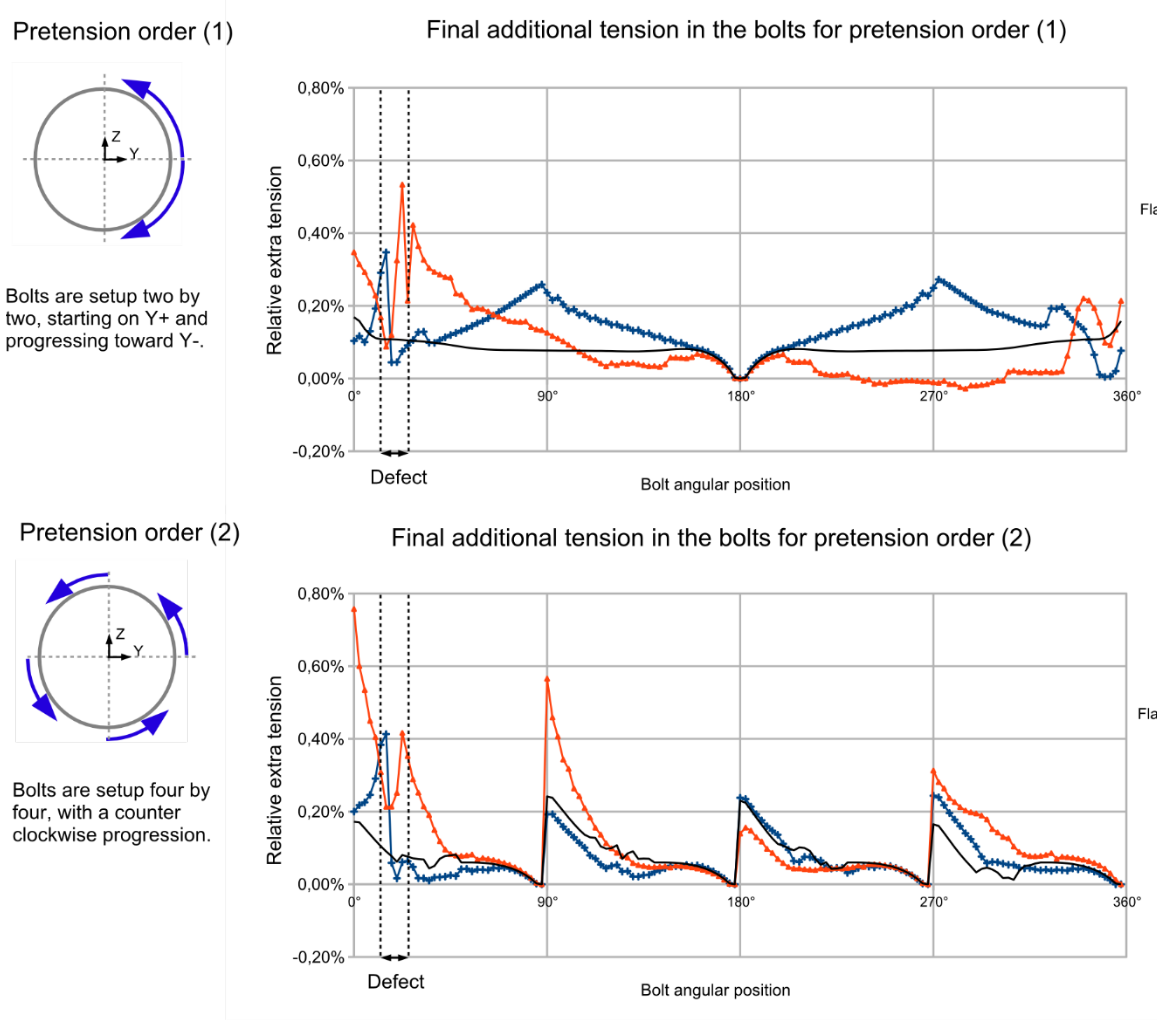


Fig. 7: Final tension in the bolt after the pretension depending on the contact geometry

Example 2: Energy dissipation by friction in the bolted joint

Friction can also be studied as a little dissipation can participate to the protection of the payload from the dynamic loads. It depends on load shape and intensity, and bolt pretension. Contact analysis under cyclic load, for a given mode shape and amplitude, can be used to estimate the corresponding damping in the bolted joint (see [1]).

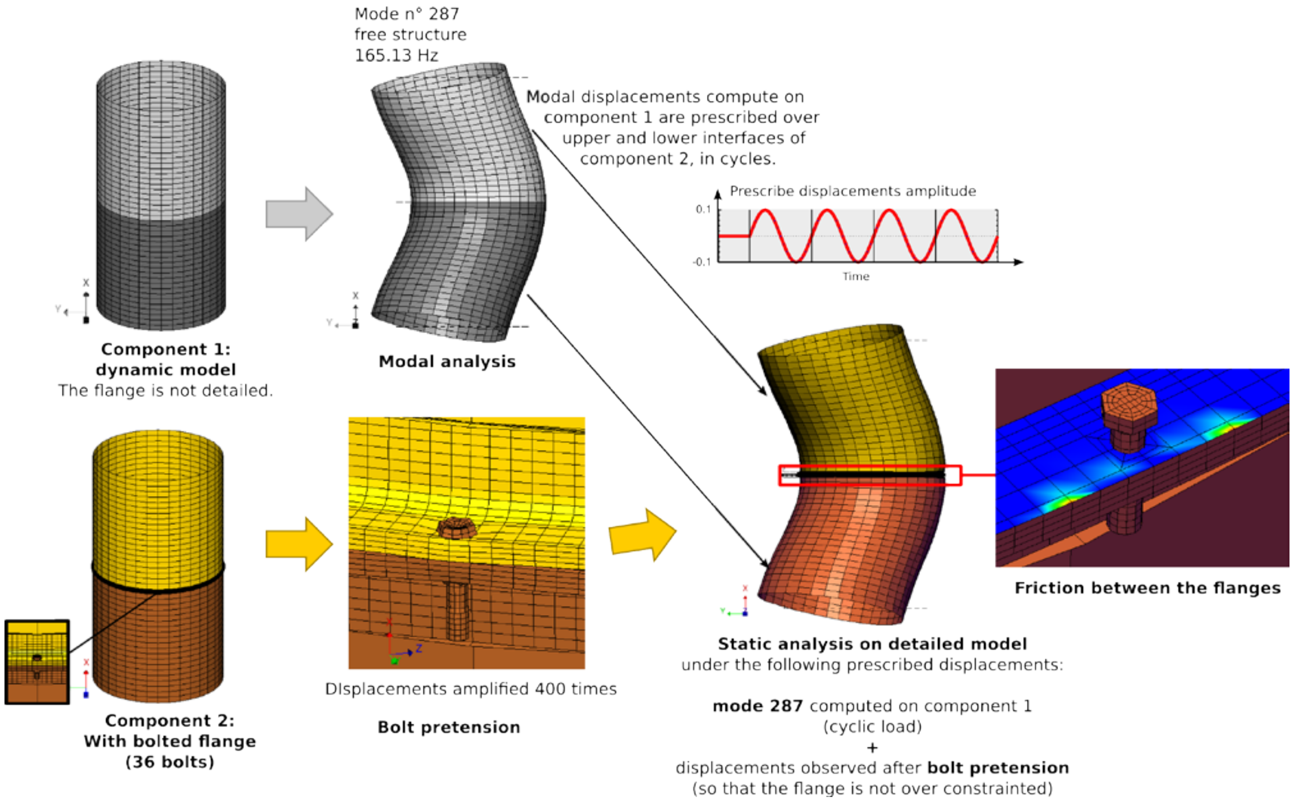


Fig. 8: Analysis of the dissipation in a bolted flange associated to a given deformation mode

5. APPLICATION TO FLUID TANKS

Fluid tanks are another example of structures with simple geometry but complex simulation data. Fig. 10 shows some non geometric data that must be considered during the modelling. In particular, the evolution of the propellant mass is considered.

This knowledge is integrated in PCGen module, as illustrated in Fig. 11 with an industrial tank model. The creation of a complete tank model for acoustic analysis, with stiffeners, inner fluid, multiple levels, inner pressurization, and basic model tests, can be completed within a few hours instead of one week.

Example: Vibration analysis

In Fig. 11, the evolution of the fluid-structure coupled eigenmodes with propellant consumption is studied. The tank model contains around fifteen fluid levels which are activated one after the other to perform a fluid-structure coupled vibration analysis. Then a short VisPER macro (written in Python language) generates pictures of each mode, for each propellant mass, so that the visual identification of the modes is fast and easy.

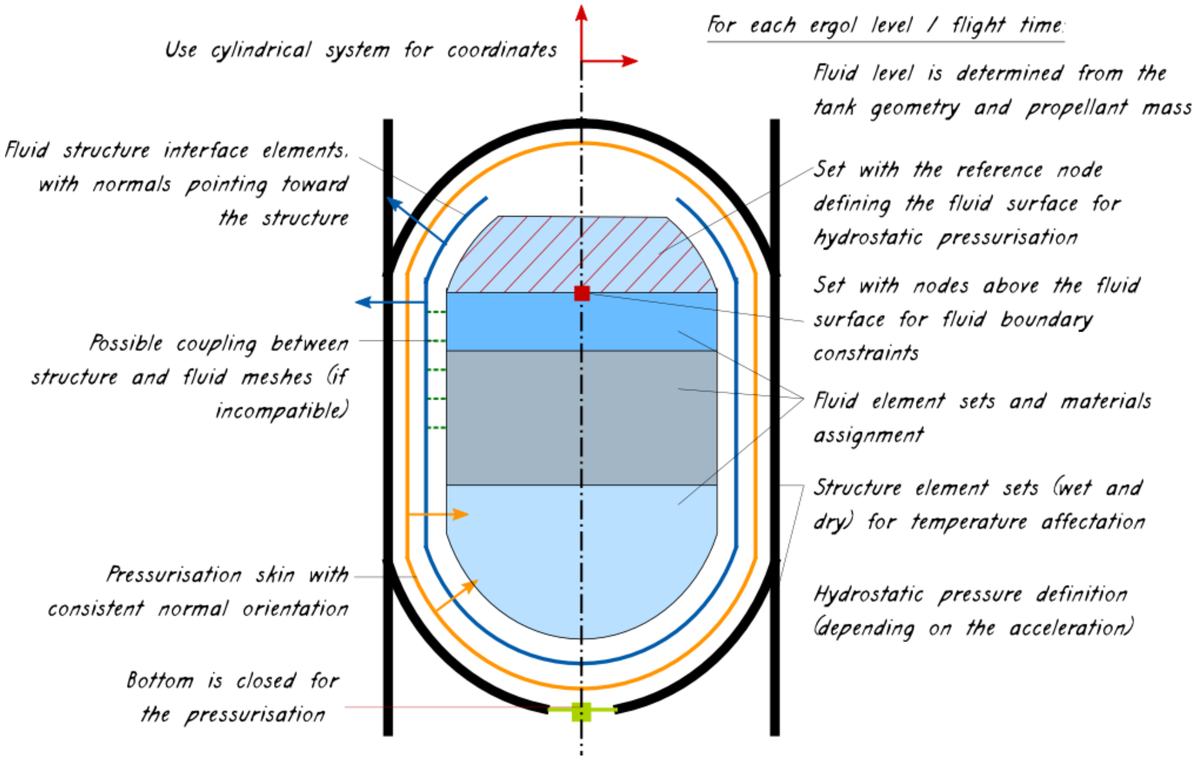
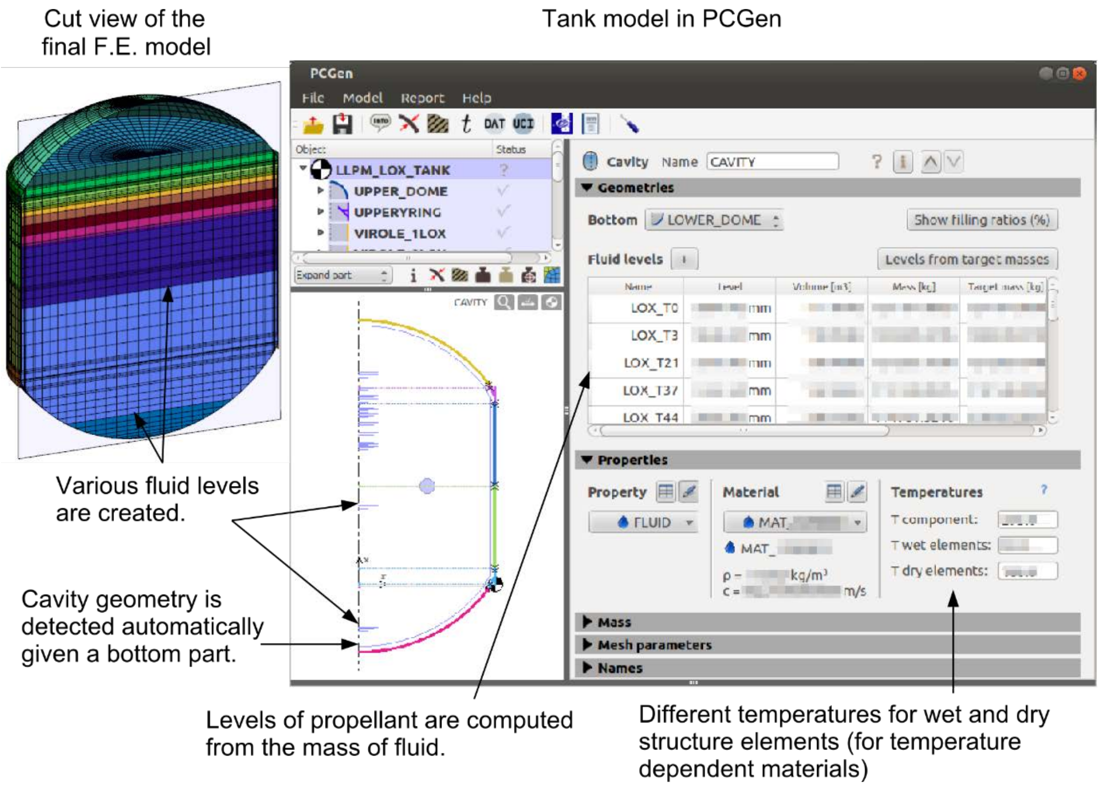


Fig. 9: Additional model data required for a fluid-structure coupled analysis



Cavity Name: CAVITY

Bottom: LOWER_DOME

Fluid levels

Name	Level	Volume [m3]	Mass [kg]	Target mass [kg]
LOX_T0	[redacted] mm	[redacted]	[redacted]	[redacted]
LOX_T3	[redacted] mm	[redacted]	[redacted]	[redacted]
LOX_T21	[redacted] mm	[redacted]	[redacted]	[redacted]
LOX_T37	[redacted] mm	[redacted]	[redacted]	[redacted]
LOX_T44	[redacted] mm	[redacted]	[redacted]	[redacted]

Property: FLUID

Material: MAT_[redacted]

MAT_[redacted]

ρ = [redacted] kg/m³

c = [redacted] m/s

Temperatures

T component: [redacted]

T wet elements: [redacted]

T dry elements: [redacted]

Fig. 10: Multiple levels of propellant are considered for the tank model

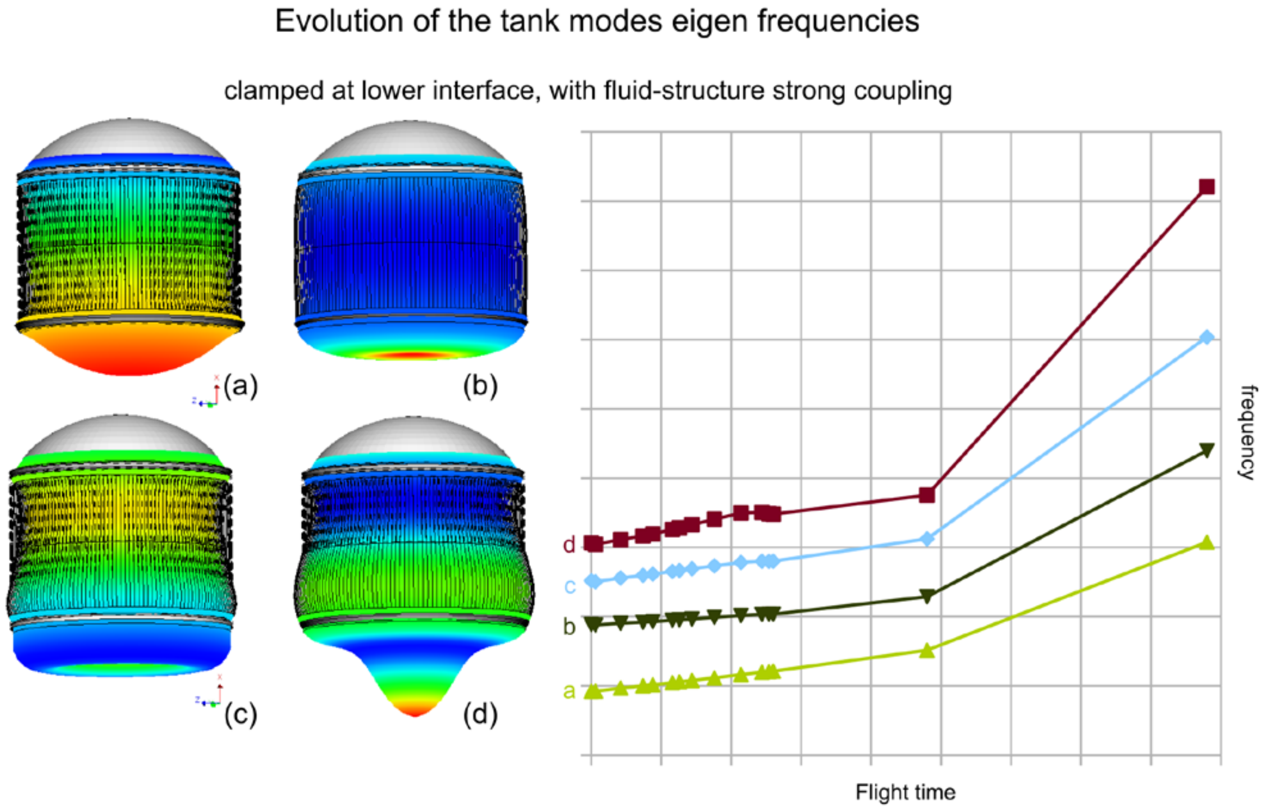


Fig. 11: Frequency evolution of some particular modes (so called “O1”, “O2”, ...) along flight time and propellant consumption

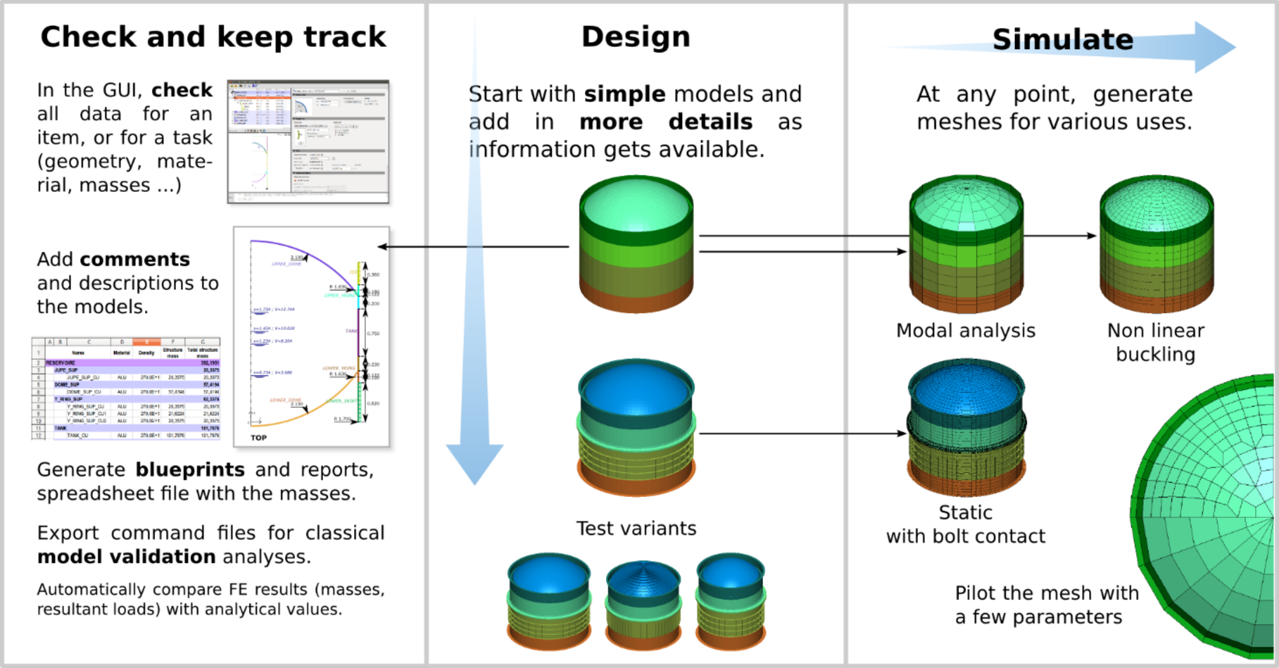
6. CONCLUSION

In this paper, we presented a new, global approach of the FE. model creation, based on a parameterization of the geometry. CAD data are no longer required, instead, the model generation is based directly on the design parameters.

Here, the various objects that build a geometry are associated with one or more specific mesh algorithm. This enable to generate automatically quality meshes along with all additional sets, data, etc. required by the analysis. So the model generation can be isolated from the geometry definition, making the modelling more flexible. By changing a few mesh parameters, various FE models can be exported for various kind of analysis. In the same time, geometry may be changed as the design evolves, the recreation of the mesh is automatic.

Moreover, since the design parameters are known, reports and blueprints can be exported with the model, (not simply geometry blueprint, but also illustrations of the material assignment, shell thicknesses, etc.). This workflow is illustrated on Fig. 12.

Finally, this process is completed by the efficiency of the post-processing in VisPER, taking advantage of macro recording for example.



In the GUI, **check** all data for an item, or for a task (geometry, material, masses ...)

Add **comments** and descriptions to the models.

Generate **blueprints** and reports, spreadsheet file with the masses.

Export command files for classical **model validation** analyses.

Automatically compare FE results (masses, resultant loads) with analytical values.

Start with **simple** models and add in **more details** as information gets available.

At any point, generate meshes for various uses.

Fig. 12: Workflow with PCGen

REFERENCES

[1] Caignot, Alain, *Prédiction par essais virtuels de l'amortissement dans les structures spatiales*, manuel de thèse de doctorat de l'Ecole Normale Supérieure de Cachan, juin 2009.

[2] PERMAS Version 16 User's Reference Manual, INTES Publication No. 450, Stuttgart 2016.

[3] VisPER Version 5 User's Reference Manual, INTES Publication No. 470, Stuttgart 2016.

[4] PCGen Version 1 Quick Reference, INTES France publication, Rambouillet 2016.