Mature Concepts Through Integrated CAD and CAE Processes in CATIA V5 with LsDyna

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Summary:

An important design case in automotive industry is the modification design. Starting from existent modules, new concepts are evolved, optimized and rated against competitor solutions. This requires a strong coupling of CAD based design with FE based simulation and a fast but nevertheless accurate simulation process. This paper shows how the coupling can be achieved up to a degree, where CAD and FE are no longer separated at all.

Keywords:

Integrating CAD FE, stamping, parametric design

1 Design Cases

In the automotive industry, two leading design cases are encountered:
- Completely new design, starting from scratch
- Modification design

The first case seems to be very common at a first glance, but it is nearly never used completely. Even exciting prototypes with high public interest share well approved components from their more conventional rivals under the hood. Instead the second case is the one which occurs most often: Starting from existing platforms, existing modules are optimized, new concepts are integrated, and these changed designs are then reused.

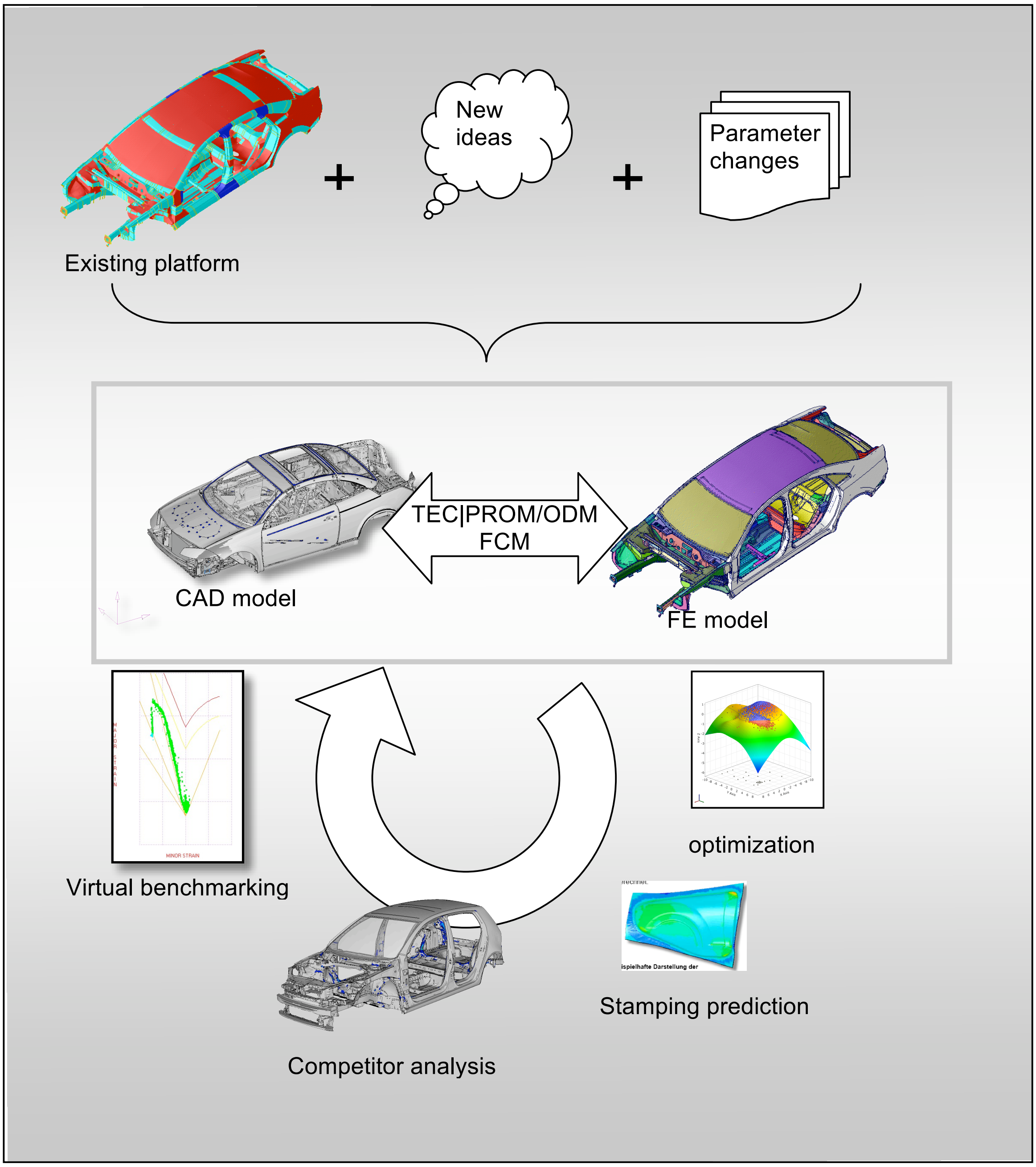
Because the re-usage of modular assemblies has become so important today, a new type of CAD engine has been developed: The *parametric* CATIA V5. Parametric means that certain well chosen variables are part of the design space. Changing the value of some parameter triggers all necessary updates and leaves the model in a valid state. Without parametric support, changing e.g. some characteristic length, leaves all the related work to the designer itself, and until his work is finished, the model is, at least to some extend, not valid for other usage.

However, modification is more than just trying out new geometry, and while the parametric concept leads to a very valuable speedup for the CAD design itself, other important parts of the modification design process are still somewhat behind:

Modification design means:
- Investigation of existing modules
- Incorporate new concepts and ideas
- Parametric changes
- FE base Optimization: structural behavior, weight, material …

The first two items describe the “classical” CAD based modification process, where existing geometry is used as a starting point for further development. Parametric changes are relatively new, as mentioned above, but their acceptance is growing very fast. While it is not trivial to build and to work with a parametric model in native CATIA, we will present a new unique tool, which makes fast parametric concept modeling very easy.

The last item is again a classic one, the FE based simulation and optimization. Although this step is of course directly related to the modification design, in many, if not most cases, it is not well integrated. The main part of this paper will discuss a new approach of better integrating the FE method and classical CAD.



Picture 1: Process overview of a design modification

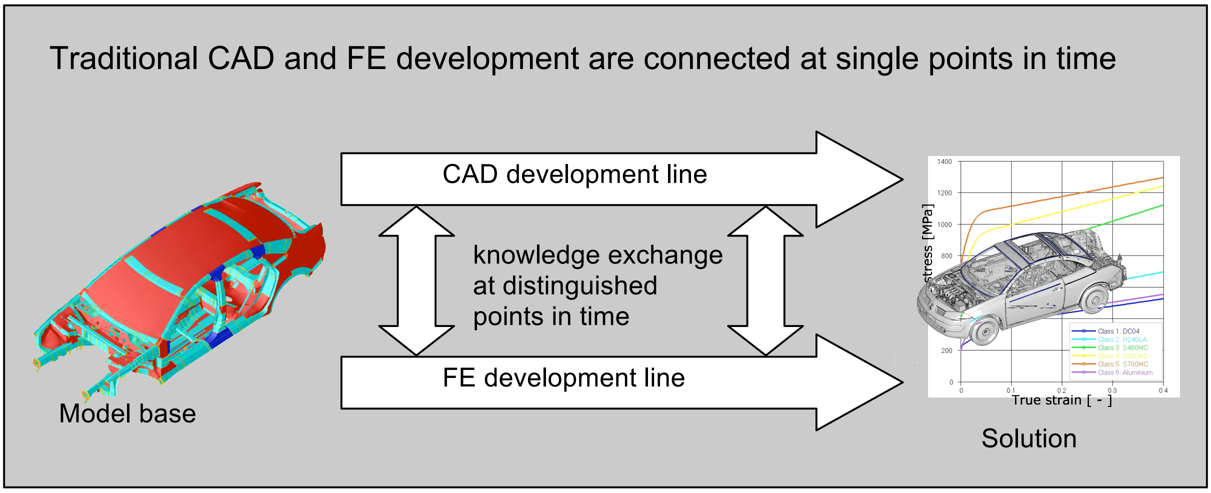
2 Current State of Integrating CATIA based CAD and FE models

Historically, development in the Automotive Industry was completely base on CAD and hardware testing. Then in the last decades simulation techniques base on FE Models have become more and more important, and are today not only an indispensable tool, but they influence the whole process at the very first steps, even at the design stage. FE simulation is not only used to check a given state of design against e.g. compulsory rules, which was among one its first uses, but it is used more and more for optimization, which has changed its role away from a pure checking tool, to an integral part of the design process.

In fact, CAD based development and FE base simulation are not simply related to each other, but it is safe to say that they are interwoven. Changes in the CAD geometry affect directly the FE mesh, changes in the bill of material need a change of the FE material card etc. Of course the reverse is also true: The result of a FE simulation may directly lead to a change of the underlying CAD geometry, it may show that other material properties are needed, it may suggest a different way of connecting parts, for example with a different spotweld density.

However, it is surprising that there still exists no commonly agreed strategy to connect both worlds.

Ideally, one would expect that at any stage during the development process, it would be possible to launch any desired FE simulations without difficulty, and that it would be possible to immediately propagate a simulation result back to CAD. But usually, both CAD and FE do indeed run in parallel, but only with distinguished points at which an exchange of knowledge is possible. The reason for such a behavior is neither a limitation of the CAD stream, nor a limitation of the FE stream, but a lack of a fast and easy connection between both.



Picture 2: Traditional CAD and FE integration

It is common to think of the CAD process line and the FE process line as two worlds, or streams, which, although connected, are both more or less independent from each other. The main purpose of this paper is to show a new concept, which will finally render the distinction between these two development lines meaningless.

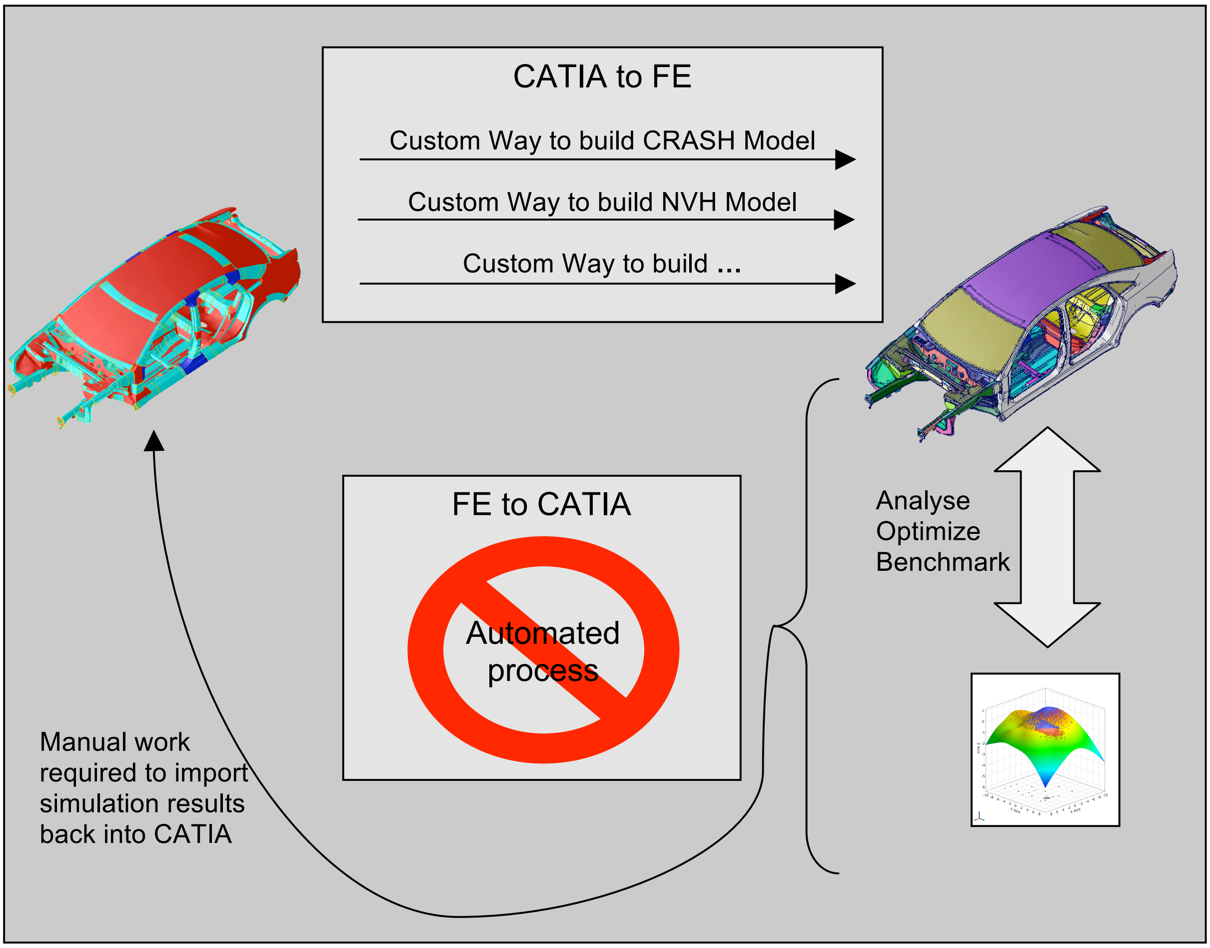
2.1 The current CAD to FE bridge

Before we show the way the TECOSIM GmbH and the ForceFive AG like to overcome the above discussed connection issue between CAD and FE models, it is necessary to analyze the current state of art, and why it seems to be so difficult. There are two different paths we must follow: The CAD to FE transition, and of course the reverse FE to CAD model transition. Both have their own requirements.

The usual way today to build a FE model from a CATIA based CAD model starts with the geometry export! This geometry is then imported into external tools, where the single mesh or different meshes are created.

Of course the FE model needs much more information then just the mesh, ref. to the following section for more details. But there exists no commonly agreed way how this information can be shared among the different solvers, external tools and the CATIA core database, and thus the result is often a whole group of different tool chains, which is needed to assemble all the required FE input.

Even worse is the state of the reverse path: *There exists no clean method at all, to import changes from an FE optimization back into CATIA.* Most, if not all changed values and changed structural details must be manually converted back.



Picture 3: Current state of the CATIA – FE – CATIA process loop

3 The TEC|PROM/ODM and FCM module in CATIA

To overcome these limitations, the Tecosim GmbH and the ForceFive AG have started a close collaboration, and can now present the TEC|PROM/ODM – FCM tool. In a nutshell, the complete FE model for whatever simulation is directly exported from CATIA.

It should be clear that such an approach immediately solves all of the above mentioned issues. When the creation of a simulation from within CATIA is easy enough, then there is no longer any need to perform changes outside of CATIA. Instead, simulation and optimization are done with the same base model, which is used for CAD. Geometric changes must no longer be re-imported into the model base from other tools. Property and material are part of the base as well as the FE model, and huge Excel sheets with hundredth of entries connecting part ids with material ids with FE ids, should no longer threaten anyone.

3.1 How it works

Even with TEC|PROM/ODM - FCM, a separate FE model as input for the simulation solvers is still needed. And this FE model will contain data which is not directly stored in CATIA, but which is based on the knowledge stored in the CATIA model, and based on plug-in databases, whose structure is provided by TECOSIM, as well as full featured data sets, but which can easily be adopted to custom needs and requirements.

The way TEC|PROM/ODM - FCM works can best be seen, if the information which is stored in the CAD database is compared to the content of an average FE model:

Common contents of a CAD model:
- The (parametric) geometry
- Part info
  - Names and Ids
  - Assembly tree
  - …
- The BOM (bill of material)
  - The thickness of parts
  - Association of parts with materials
  - The name of the material to use
  - Material data: the Name, Youngs modulus, density …
  - Other material data such as stamping information etc
- Connection data
  - Spotweld positions
  - Spotweld data, such as thickness
  - Glue/Seam line data
  - …
- Special load data for non-standard load cases
- …

This (incomplete) list is in general independent of any required simulation load cases, and as such cannot contain all the required knowledge which is needed to build a FE model. A closer lock at the content of an average FE model makes this clear:

Content of and required knowledge needed to build a FE model:
- The solver to use
- The mesh
- The part info
  - Thickness per part
  - Material per part
  - Other properties like hourglass information
  - Stamping data, as material preload
  - …
- Material database

- Connections
  - Spotwelds
  - Glue and seam lines
- Load case(s)
  - Type of load case
  - Actual load data (forces, moments, acceleration, …)
  - Load case dependent extra data for standard cases
    - Dummies, Barriers
    - Other test specimen
- Result capturing
  - Measurement springs, Accelerometers
  - Time history data
- …

Both lists mention only the most common data. It is immediately clear that some items have a simple one-to-one relationship, for example the part thickness, however, for most others the relation is more complex. Of course the most non-CAD based influence on the generated FE model is the type of the simulation. Whether it is crash, NVH, durability or any other case, it strongly influences the FE model. To be more exact, there exists a dependency on the type of simulation (Crash, NVH, …), and within a certain type both the chosen solver and the load case (e.g. Rear Crash or ODB) might determine even more fine grained differences:

Influence of the Simulation Type and Load Case on the FE model:
- The available solvers (and hence the FE input format) are narrowed by the simulation type
- FE materials are strongly load case dependent (static-elastic, explicit, durability …)
- Different meshes with different quality are used for different simulation types
- It is still an open debate whether different load cases within the same simulation type require a change of the mesh
- Connection modeling is both solver and simulation type dependent
- Instrumentation for time-history result capturing is simulation dependent
- … (there are more dependencies, of course)

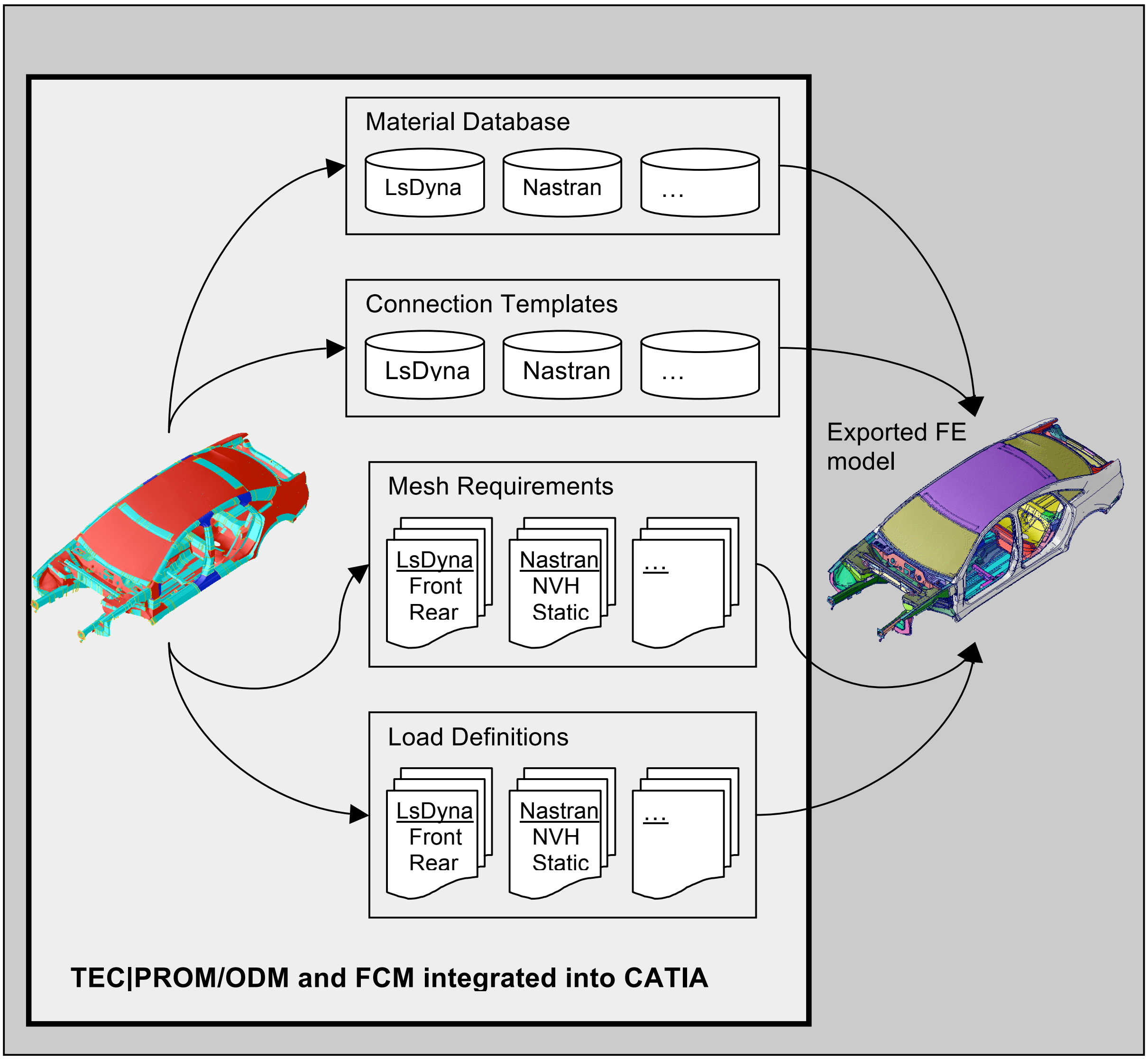
The above list shows clearly, that it does not make much sense to store information about all *possible* simulations into the CAD model. Instead, another approach was chosen:

the TEC|PROM/ODM - FCM tool uses the concept of plug-in databases for different simulations

A crash is a crash is a crash, which means that the required additional info is always the same: Some barrier is needed, the initial velocity is determined by the crash type, as are the mesh requirements and the time history requirements. Thus the TECOSIM GmbH has assembled a database, which holds all needed barriers for both the common crash types and the common solvers, and another database holds the load data, which in this case is simply the initial velocity of the vehicle.

Similar databases have been build, or are currently filled, for other simulation types. However, a major point while developing the concept, was *extensibility and customization.* Tecosim and ForceFive are well aware of the fact, that there is always a need for special customer dependent additions. A simple example might be a nodal mass distribution for crash, which compensates for neglected vehicle parts, and might depend on the BIW and the engine used for the current model. A set of such distributions can easily be added to a custom database, e.g. in the most simple form of different include files, whose inclusion is triggered e.g. by the presence of a certain engine assembly.

Utilizing this strategy, a customer can use the provided databases as long as they suit his needs, but he is not limited by hardwired boundaries. And even if more and more specialized simulation programs appear on the market, TECOSIM and Force Five strongly believe that they can be handled as well.

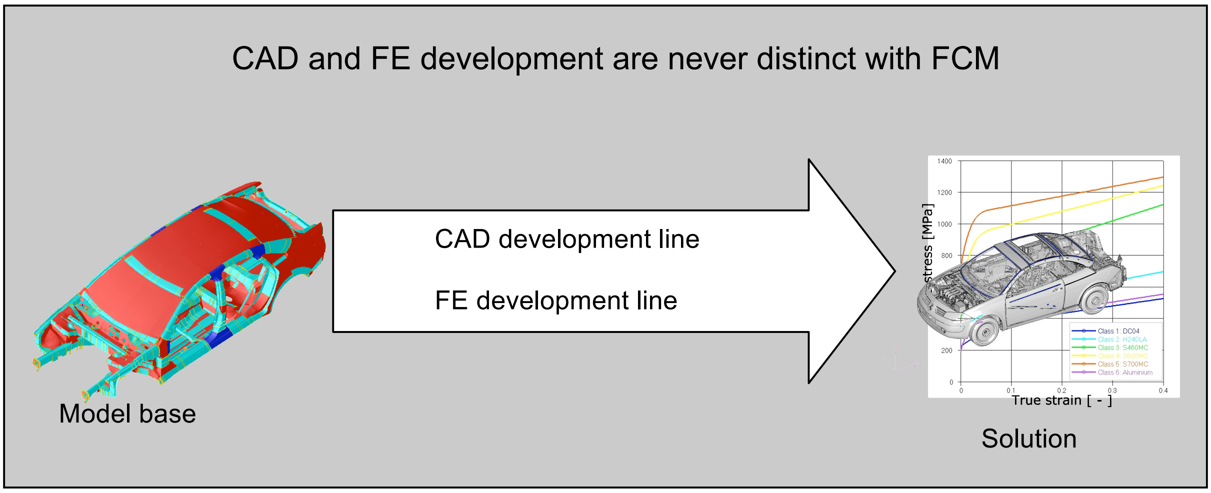


Picture 4: The Fast Concept Modeling (FCM) tool

The development of the TEC|PROM/ODM - FCM tool is ongoing at a very fast pace, and we expect to present some real world examples within the next few month.

> *Working with TEC|PROM/ODM - FCM means that all design changes are done within CATIA. There is no need for a huge list of external tools, there is no need for custom tool chains, and there is no need to manually re-import changed structures into the CAD model. As a result, the design process will be faster, more reliable and more efficient.*

If we compare this status with the picture 2, we can see that there exists no separate FE design stream, but both development lines are integrated into each other.



Picture 5: Integrated CAD and FE design development with FCM

3.2 Working showcase with TEC|PROM/ODM - FCM

The current state of the FCM tool is more than a concept. Tecosim and ForceFive have been working on the implementation for some time now, and we can present a showcase which is only short of becoming a real-world scenario.

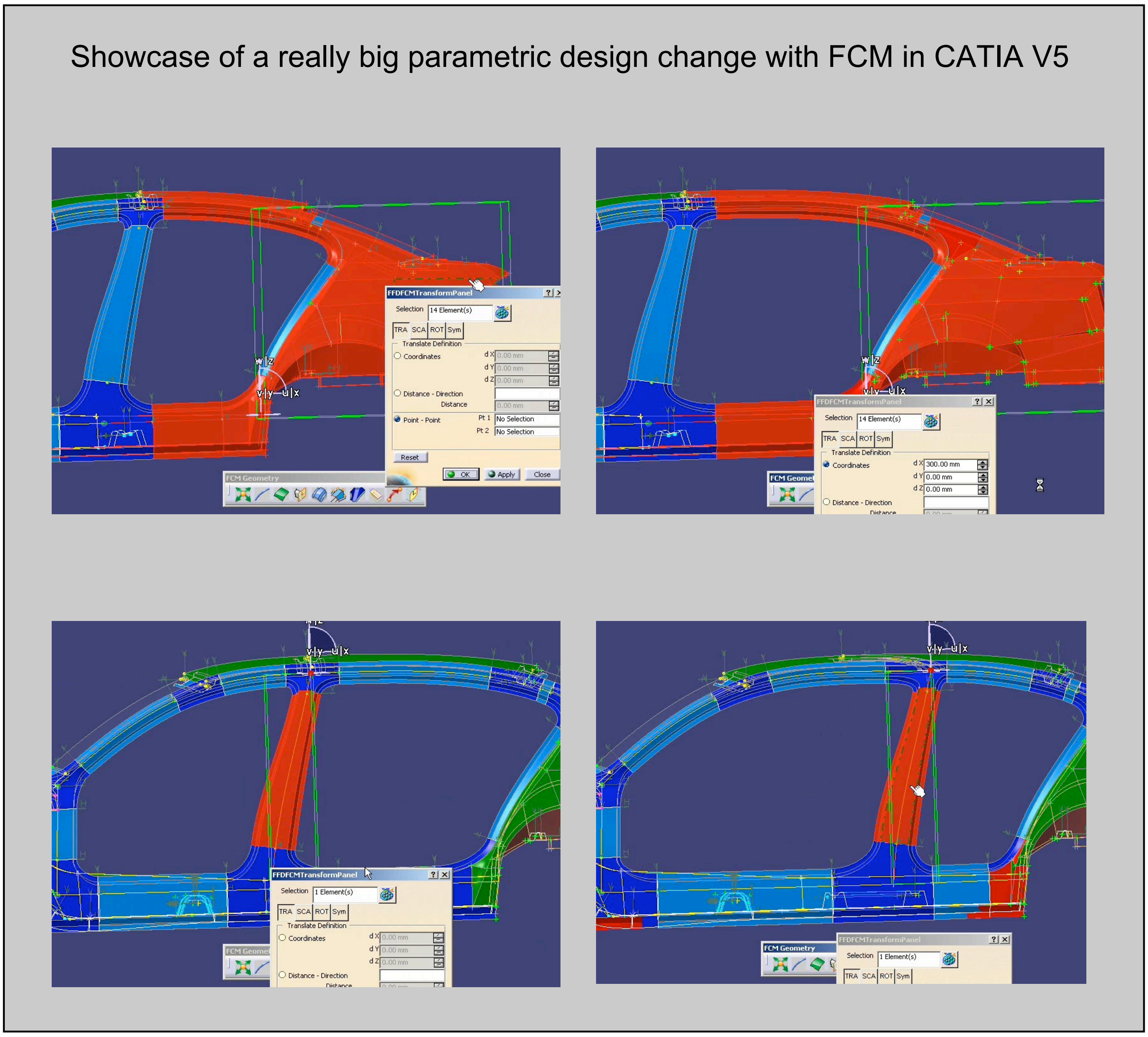
3.2.1 Parametric study with FCM

As mentioned in chapter one, TEC|PROM/ODM - FCM not only provides the integration between CAD and FE, but its FCM module also supports the concept of parametric modeling with a new interface which is especially suited for fast FE concept studies.

The FCM module is developed by the ForceFive AG. It would be out of scope of this meeting to give a full presentation here, but the following two screenshots of a parametric design change in CATIA with FCM will at least clarify the power and potential of this approach. (The live presentation will include a short session movie, which proves that even the big changes done here are really fast)

The screen shots show two successive changes to a body-in-white: The upper left picture shows the unmodified geometry, but with the rear end already selected for a change. The upper right picture shows the result of stretching the rear end. The stretching itself is done by interactively selecting a stretch direction and amount. It cannot be seen from just these pictures, that the FCM tool (and the underlying parametric model) take care of anything: All attached geometry is handled with ease.

The 2nd row of the screen shots shows a subsequent shift of the B-pillar. Again, just selecting the central part of the pillar, shown in read, is sufficient. Both changes takes only a few seconds to complete, which is rather fast.



Picture 6: Showcase of a really big parametric design change with FCM in CATIA

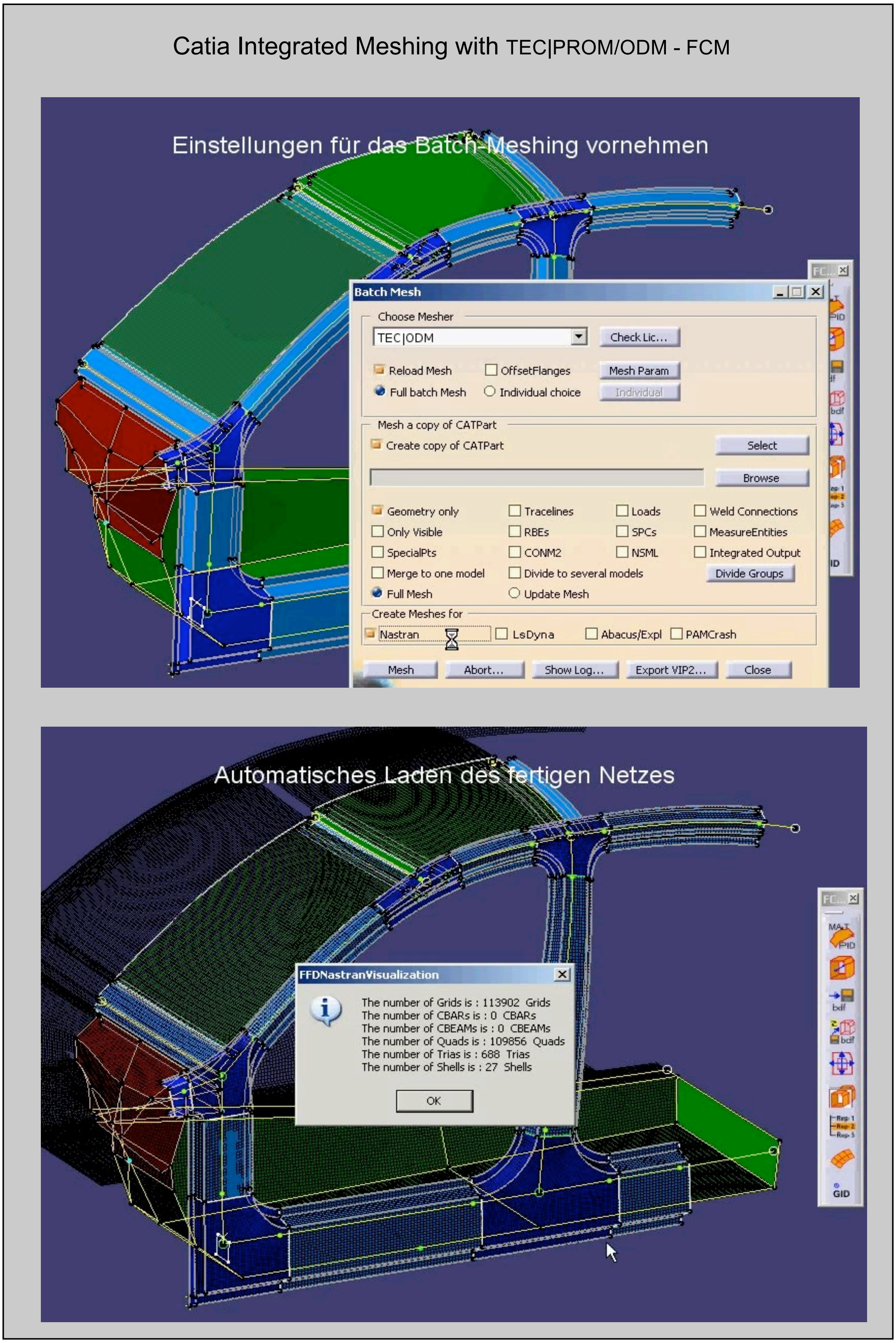
3.2.2 Creation of a FE model with FCM

After performing any designated design changes, a FE Model must be created, and the simulation can be started. Creating a FE model can now be as simple as a few mouse clicks, as is shown in the pictures below.

The backend for meshing, which is used in the TEC|PROM/ODM - FCM tool is not the native CATIA mesher, but based on the well known TEC|ODM meshing kernel. This means, that high quality meshes can be generated, which are suitable for all kind of simulations, including explicit ones, of course.

For those who already have experience with TEC|ODM, it seems appropriate to note that the meshing kernel has a new *reduce Triangle* algorithm added. The Tecosim knows that the number of triangles in the final meshes was a former point of criticism, but this is now completely obsolete. The meshes generated by the current TEC|ODM have reached a quality, which is considered best in class.

The following screenshots show how easy it is, to create a read-to-run FE deck from within TEC|PROM/ODM - FCM.

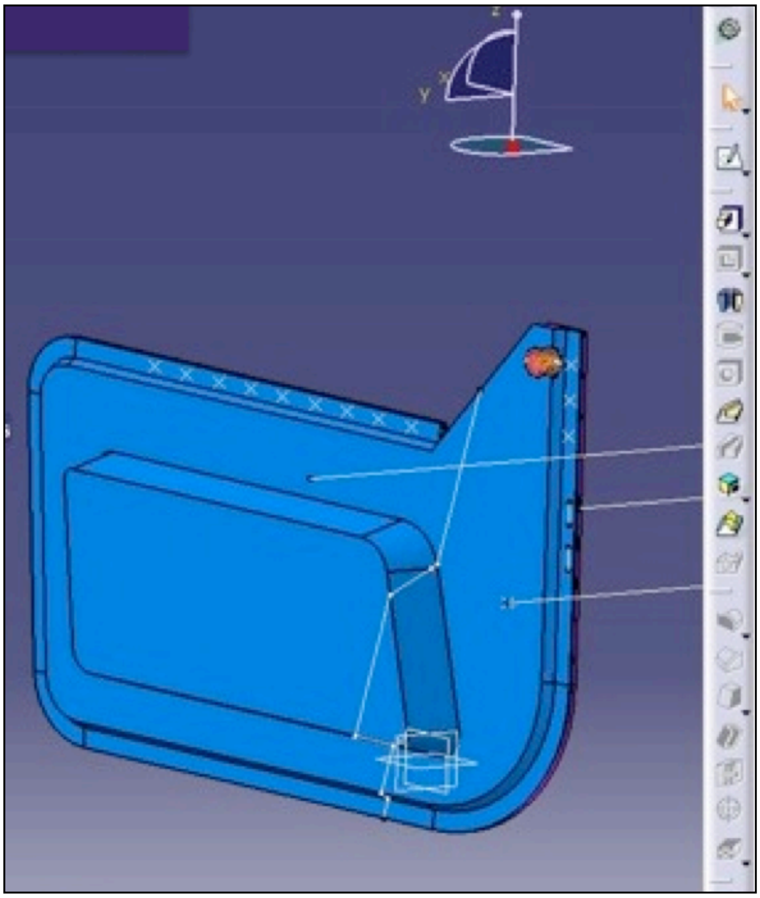


Picture 7: CATIA V5 integrated meshing with TEC|ODM

3.2.3 *Definition of special FE constructs in CATIA*

As mentioned above, the content of a FE input deck may contain data, which is not only CAD based, and which cannot be deduced from the type of simulation and load case. Examples are special loads on points or surfaces, and measurement instrumentation for time-history output.

All of these can easily added within the CATIA base model! The following screen shot shows a part of a vehicle door, which has special load sets added, where each load set may consist of several loads. This data is now part of the CATIA model tree. It should be noted that the load definitions in the example are vector quantities, the realization for the FE output depends on the chosen solver.



Picture 8: FE loads in CATIA

4 Summary

This paper has presented new but mature concepts, which will lead to a great speedup of the design modification process.

Currently CAD design and FE simulation are in principle two distinct development streams, and their coupling is matter of custom procedure chains and manually rework. The first enhancement proposed here, is a native coupling of both CAD and FE within CATIA. All design changes, whether shape, material property or whatever, are done in one base model. FE simulations are simply triggered by exporting an appropriate solver input deck from within CATIA.

The needed information for the FE solver comes from two input sources: The base model, which, aside from the geometrical data, also provides properties such as thickness values and material specifications, and a set of databases which hold solver and load case specific information such as material cards, test specimen etc.

Using the new approach will lead to both a speedup and a better quality of the design process. Results of simulation need not be propagated back to the model base, but are immediately available, and design changes can immediately be checked for implications.

This also means that the possible design space becomes much broader, because it is more easy to develop and investigate design changes. Consequently, this leads to the usage of a parametric design space, where even the shape of a part is no longer something fixed, but a range of possible shapes. A new tool, which allows easy manipulation of and working with this design space, was presented in this paper.

Combining both concepts, will not only lead to faster development cycles, but it will open up new opportunities: Using optimization techniques over a parametric design space, allows the investigation of concepts, which cannot be handled the traditional way.

