FINAL ELEMENTS METHODS AND THEIR UTILIZATION IN THE AUTOMOTIVE INDUSTRY.

Július Roób – Ján Piľa

This article introduces the most common methods based on the CAD and FEM. As well, briefly explains the principles of the model preparations, analysis set up and the ways of solvers calculation. Also shows a few examples of their implementation right into the automotive industry research and production.

1 INTRODUCTION

As well as in every technical area, there is visible a growing effort in automotive industry to implement new, progressive methods right into the process of structural design and the production itself. If we consider the process of structural design development to be a search for optimal alternative, than the utilization of these new ways of constructions creation makes it more effective and may bring the optimal results (in compliance with all requirements) in shorter time period and with less expenses. These methods are basically divided into two major sections. The first group consists of CAD methods (computer aided design). Those are primary designed for the modeling of three dimensional structural components and systems from the point of their shape and geometry properties. In fact, they used to be the first step for designers and constructors. Their output (modeled geometry) may become an input for the second group so-called CAE (computer aided engineering) which is mainly based on FEM (final elements methods) and provides a wide range of possibilities. Still developing, this progressive method has already become very powerful and effective tool for predictions and modifications of components behavior even before the first real prototypes are prepared for tests.

2 FINAL ELEMENTS METHODS

The basic idea is quite simple and comes from the fact, that every component may be considered as a material continuum which consists from the infinite number of mass points. Naturally, it is not possible to work with or calculate the structure like this. The only solution is to replace the components volume or middle surface with the finite number of 1D, 2D or 3D elements connected trough the nodes to homogeneous mesh that corresponds with the original geometric shape. Now we are getting the mathematical model with the same material and mechanical properties as the real components has, but due to finite number of used elements the solver is now able to run and finish required analysis. In general, the more elements with lower dimensions we use the more precise results we may obtain. In this way, it is possible to model and set-up for the calculation any real components and if necessary implement them into the sub-systems and major units (Figure 1).

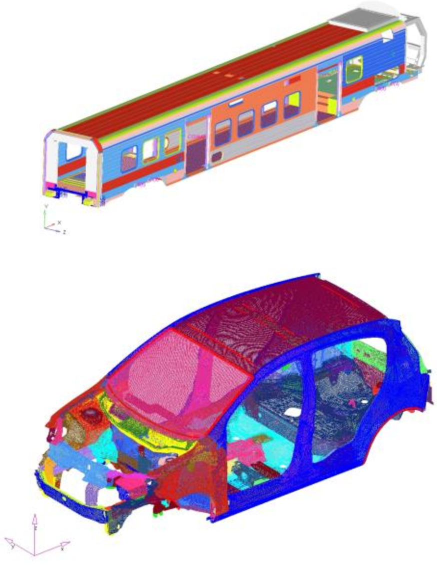

Figure 1. Examples of structural models from automotive industry created from the final elements.

Of course, in order to create the whole systems (including the vehicles of any type), it is not sufficient to know just the geometry of each component, but it also requires the knowledge about their positions, relative interactions, ways of connection etc. and simulate them with highest possible precision. It becomes even more important when the performed tasks are non-linear and includes the dynamic load cases like crash tests, mold flow calculations, fatigue analysis and others. In all of the mentioned cases, the way we describe all known boundary conditions has the significant influence on how close the simulation is able to approach to the real load distribution through the structural system. In general, linear problems based on the static loads acting are considered to be the basic and less difficult tasks. To solve them, just simple linear modules as optistruct are sufficient. Nevertheless, during the components mechanical life span the acting of non-combined static load cases are not very frequent, and in the FEM calculations we usually define them just in two cases:

1) If it is possible to neglect the changes of the acting forces in time or eventually if the constructor already has a knowledge (either from the previous experiences or it results right from the task) that the differences obtained after we change the dynamic problem to static are acceptable for the results and they are not adequate to higher time consumption which is required for the dynamic problem preparations.
2) If there is a request for the stiffness determination of the structure. In the automotive industry the final structures often has to satisfy particular standards (from the point of their mechanical properties), usually defined by the constraints based on the local and global stiffness. The objective may be for instance the simulation of local, short-time loading corresponding to the real situations which may occur as forces caused by the wing maintenance, airframe assembly operations, and moving parts impacts etc. (see Figure 2). As well, the global stiffness attestations are often required (see Figure 3). In all of the mentioned cases, just the boundary conditions and simple loads and moments are applied on the specific positions of FE model. Evaluating the stiffness, not the stress distribution is interesting but the deformations (displacements in case of forces and angles of rotation for moments) in the direction of acting load. With the results, using the simple relation between the acting force and corresponding deformation we are able to get the stiffness values (the force needed for the unit deformation). In real automotive production it may be quite an important information, because it can help to prevent the construction from irreversible plastic deformations

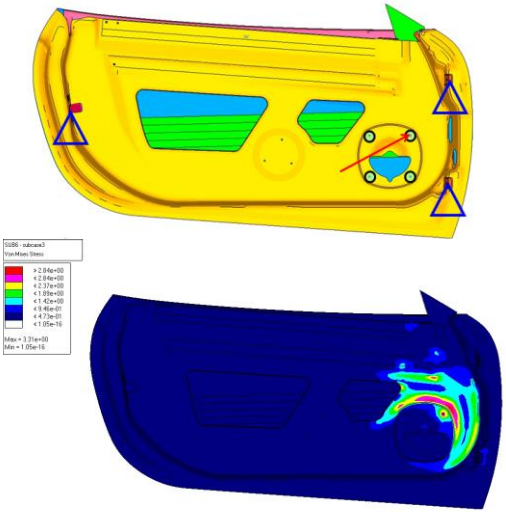

Figure 2. Local stiffness analysis performed on the internal side of the car doors in positions where the speaker are about to be mounted. Blue triangles shows the constraints, red arrow is the position of the force.

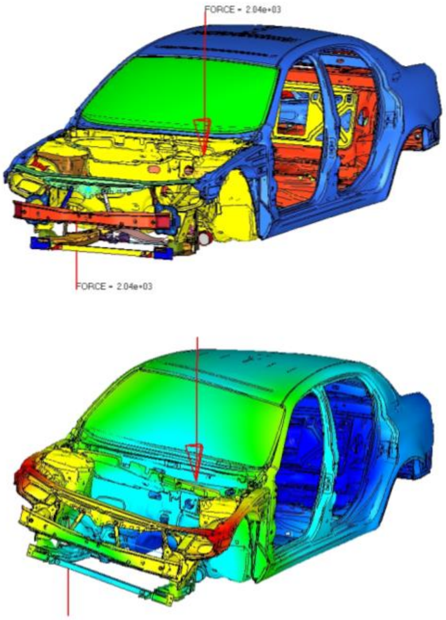

Figure 3. Global stiffness analysis of the cars chassis. The deformations scale is 1:200.

More difficult problems are presented by the situations, if it is not possible to neglect the effect of the acting forces in time. In these cases we speak about the dynamic loading, so they can not be considered as linear problems. In comparison with the basic linear static tasks, these require more precise understanding of the real situation we are going to simulate. The preparation itself becomes more difficult, even though we may use exactly the same final elements model like in linear static. The reason is that the main difference does not consists in the model, but in definition of boundary conditions taking into account the load changes in time and what is even more important, in definition of possible relative contacts between the components. As an example we can mention the impact forces during the crash tests. Nowadays it has become usual proceeding that precise virtual simulation using FE methods and non-linear modules precedes the real tests performed with the physical prototypes. Right here the set up of all boundary conditions and contacts shows its correctness, because just the deformation of the few components positioned in front of the car is caused by immediate impact to barrier. All the others parts situated beyond them depend on their deformations and displacements which are distributed continuously trough their relative contacts. From these reasons it is very important to understand the rules of simulated action as well as the environment of the dynamics solvers in order to obtain reliable results. On the other hand the utilization of these methods during the structural design creation (or modification) is very effective mainly if we consider all expenses needed for the real test.

Another separated area (although always depending on the results of basic mechanical analysis mentioned above), visibly growing in automotive industry is the structural optimization using final element methods. In general, this process helps constructors to find an optimal equilibrium between mechanical requirements (structure stiffness, deformations, maximum stress values etc.) while still having the best possible structural properties (low weight, effective structural design without useless material and consequential expenses saves etc.). And exactly the structural optimization represents the solution how to reach optimal design of the components and whole structures. Three main structural optimization disciplines, or categories, have been developed: sizing, shape and topology optimization. Their principles are shown on Figure 4.

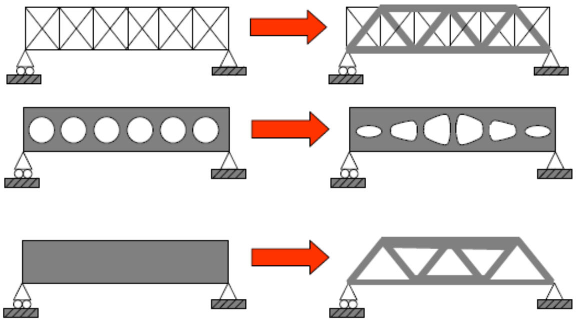

Figure 4. From up to down: size optimization, shape optimization and topology optimization

In fact, the process itself comes from quite a simple idea and requires the same finite elements model as any other analysis except that now the optimization set up has been added. It contains mainly the information about the objective (minimize mass, volume, displacements etc.) and constraints (maximum allowed displacement, stress value etc.) Than, during the calculation the solver is trying to find the optimal amount and position of the material in components volume, taking into account all load cases as well as physical and optimization constraints. On Figure 5 we can see the example – the rib of the wing with the thicknesses values suggested by the optimizer. As we can see, elements in shown in red lays right in the path of load distribution trough the model so these areas requires the highest thicknesses. Areas shown in blue are without or under the very low stress so the necessity for the material carrying acting loads decreases.

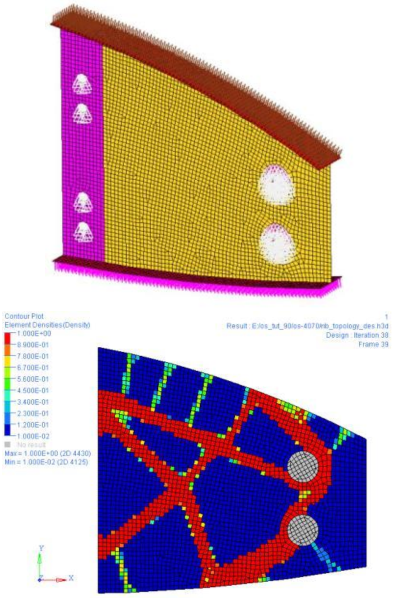

Figure 5. Topology optimization of the wings rib. Elements in red shows the areas with highest stress concentrations where the major amount of material is required.

Virtual optimization may be used mainly in two basic cases. The first and naturally less efficient way is the application with the existing structural parts in order to improve their properties. Nevertheless, in this way original components design has been already created by standard methods and implemented into the main system assembly, so the possibilities of modification are usually bounded, for instance by the connections with surrounding parts. On the other hand, optimization process may be also used at the beginning of the design process, when the component does not physically exists yet. In general, this is the better solution because constructor has much more freedom in the creation of the optimal components design right in the first phases of projection. Using the proper optimization method, it is possible to explore the material volume which has just a very rough shape of component. Than, according to the required parameters, boundary conditions and loads distribution, optimizer is able to suggest the shape very close to optimal design. It is usually done by using so-called density method, where a single continuous variable $\rho(x)$ is defined for each element included within the components design area. This variable is virtually connected with the element stiffness by the following relationship:

$$E_{ijkl}(x) = \rho(x)^p E^o_{ijkl},\ p > 1,$$
$$\int_\Omega \rho(x) d\Omega \le V; 0 \le \rho(x) \le 1.$$

In these equations x represents the position vector and E^o_{ijkl} the stiffness matrix according to the material properties. The term $\rho(x)$ is usually presented as density, mainly if consider that the volume is calculated as $\int_\Omega \rho(x) d\Omega$. And because we can say that $E_{ijkl}(x, \rho = 0) = 0$ as well as $E_{ijkl}(x, \rho = 1) = E^o_{ijkl}$, the density control the existence of each element. So it means, that optimizer evaluate the necessity of each element and decide whether its presents has the significant meaning for the loads distribution trough the volume. According to this, the values between 0 and 1 are assigned for every element where 0 means that no material is needed and 1 requires the full density of material to carry the acting loads.

As an example we can mention door support arm of the airliner. In order to decrease the weight and remove redundant material, chosen components including this arm has been optimized. After the application of the topology optimization and all necessary geometry modifications according to the optimizer suggestions the final shape has been received (see Figure 6). The new design, while still passing all stiffness and deformations requirements has lost almost 20% of its original weight.

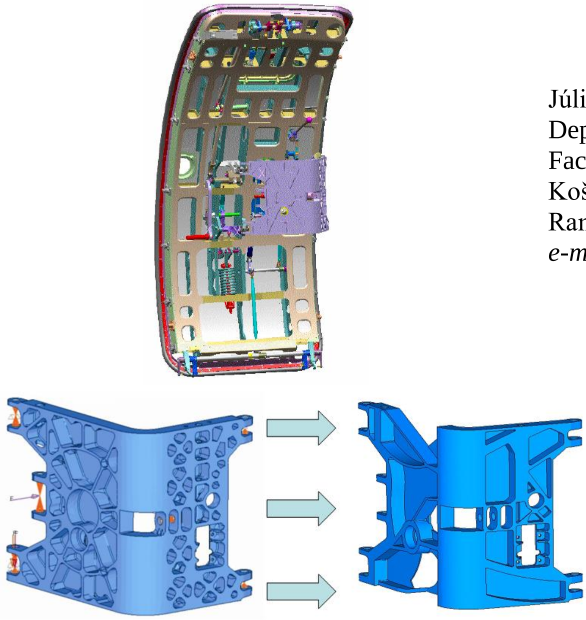

Figure 6. Topology optimization of the door support arm. Component in the middle is the one with the original design, on the right is the same part after the optimization process.

CONCLUSION

As we can see, the techniques of structural design projection mentioned above are able to make the design process and virtual testing easier and more effective. Their utilization may significantly decrease the time consumption and economic expenses along with the same or even increasing mechanical properties of the components and material saves. A lighter structure positively affects the parameters of all vehicles, what is mainly visible in the aircraft industry. Proceedings and possibilities related to FE methods are wide and wise usage may eliminate time wasting, iterative process based on tries and mistakes.

REFERENCES

[1] Altair HyperWorks Optistruct tutorial

AUTHORS´ ADRESSES

Július Roób, Ing.
Department of Aviation Engineering
Faculty of Aeronautics Technical University in Košice
Rampová 7, 041 21 Košice, Slovakia
e-mail: julius.roob@amet.it