Petru DUMITRACHE

ABOUT EFFICIENT MODELLING OF THE GEOMETRY FOR FINITE ELEMENT ANALYSIS

Abstract: *The current CAD platforms provides to user a wide variety of powerful geometric modelling tools, which allows the generation with a high accuracy the 3D geometry of the machinery components. If for achieving the fabrication documentation, the accuracy of the geometry is undoubtedly indispensable, for finite element analysis this approach of the geometry modelling is not always recommended. In this context, after a brief presentation of the general framework of the issues approached, the article highlights the main aspects to be considered in modelling of the machinery and construction components geometry for efficient study of their behaviours using finite element analysis. Also is introduced the concept of minimal geometry as the foundation of efficient modelling of the geometry for finite element analysis.*
Next, a case study is presented, case study in which geometric modelling has followed the aspects enunciated. In final part of the article are presented some comments and conclusions.

Key words: *Efficient modelling of the geometry, minimal geometry, finite element analysis.*

1. PRELIMINARY

The upward and accelerated trend of the hardware performance in decades has provided support and created the premises of increasing complexity of the software applications that have become true work platforms, concurrently with considerable broadening of the problems field.

In mechanical engineering, recent decades have imposed the computer aided design and the finite element analysis.

In that effect, CAD platforms provides the foundation for the generation of virtual mechanical systems and FEA platforms allow depth study and analysis of mechanical systems in the most various conditions with high productivity, unsuspected now several decades.

In this context, the CAD and FEA platforms have become essential for modern engineer in its attempt to design products as close to the desired optimum.

Usually, FEA platforms have internal geometric engines in order geometry modelling, which offer the user different tools that allow modelling of geometries, more or less complex. The development and extensive use of computer aided design platforms, having a wide variety of extremely powerful tools for geometric modelling, led to an important reorientation of FEA software producers. In this context, the vast majority of existing FEA platforms has relatively modest geometric modelling tools, as in compensation, able to recognize and import geometric models created on external CAD platforms.

Typically, studying the behaviour of a mechanical system using finite element analysis involves the following stages:

- analysis of the real mechanical system and associated physical problem and setting of the associated relevant results;
- generating a geometric model associated to the target mechanical system;
- generating a finite element model associated to the real mechanical system based on both geometric model previously generated and relevant results;
- generating the numerical model associated to the finite element model;
- solving the numerical model;
- analysis, interpretation and validation of results.

The diagram shown in Fig. 1 reveals the succession of stages listed above, as well as interconnections between them.

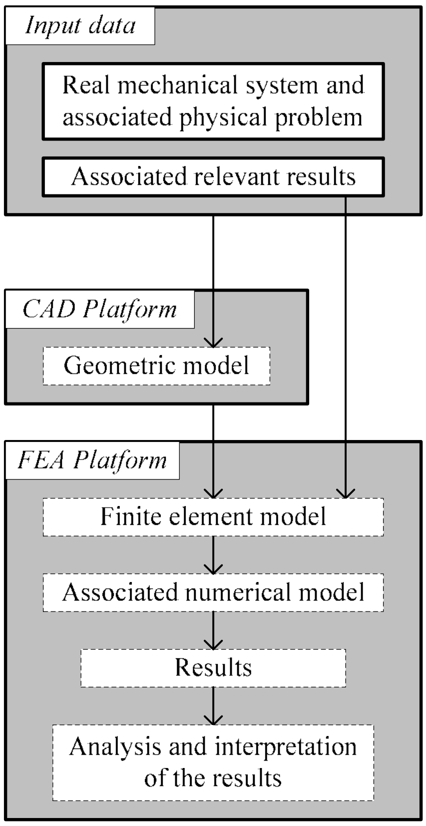

Fig. 1 – Stages of the behaviour study for a mechanical system using finite element analysis

In connection with the diagram in Fig. 1 it specifies that for a simple real system, finite element model is obtained directly, without a prior geometry modelling.

Except as simple real systems, in fact a rarely seen case in engineering practice, the geometric modelling of the mechanical systems has a particular importance in the sequential process developed between the real physical problem and results obtained on basis of finite element method. Simultaneously, geometric modelling being an early stage of the process, prior modelling with finite elements, manner of geometric modelling can have overwhelming influence on the process of finite element modelling and hence over the type of the obtained results and their precision.

2. ASPECTS OF EFFICIENT MODELLING OF THE GEOMETRY FOR FINITE ELEMENT ANALYSIS

The essential purpose of the finite element analysis on a real mechanical system is to validate its functionality under the imposed conditions.

Consequently, obtaining results with a good accuracy is a main target of any finite element model. Otherwise, finite element modelling leads to unjustified increasing of the project completion time and may be the basis on false conclusions (and frequently dangerous) about the studied system.

Based on the above, we define *optimal finite element model* as the finite element model that ensures results with the imposed accuracy.

Since finite element model is generated on basis of the studied system geometry, it needs to contain only those geometric elements that lead to an optimal finite element model.

Thus, it is introduced the concept of *minimal geometry*.

In this context, minimal geometry of a real mechanical system is its associated geometric model containing only the *necessary* and *sufficient* geometric elements capable of leading to optimal finite element model.

Obtaining minimal geometry is in our view one of the main objectives to be considered by the analyst engineer.

In fact, minimal geometry associated to a real system is an approximate or simplified geometry of his real or complete geometry. This is why the process to obtaining the minimal geometry is often called *geometry simplifying* or *geometry disfeaturing*.

Introducing of the stage of minimal geometry modelling associated to a real system, as a compulsory stage, its manufacture is done only after its validation based on analysis of its behaviour under the conditions imposed by the project. Consequently, the diagram shown in Fig. 1 should be completed accordingly. The diagram in Fig. 2 shows the precursory stages of the production of a mechanical system.

The modelling of the minimal geometry must be approached with maximum attention, because this is not a mere simplification of the complete geometry of the studied system.

As a result, minimal geometric model of a mechanical system will be linked and will take into account a number of issues, of which the most important are below presented.

The nature of the problem studied. It correlates the geometric modelling with the requirements imposed by the real physical problem and by hypothesis of solid schematization accepted in scientific field of the problem (e.g. Strength of Materials). If, for example, in case a plane stress state problem is not strictly necessary for the 3D geometry of the structure. In this case only the 2D model is necessary, in plane of the stresses studied.

Similarly, where the structural element can be assimilated with a plate, it is desirable that the minimal geometry to be represented only in the median plane of the plate which is associated with the structural element.

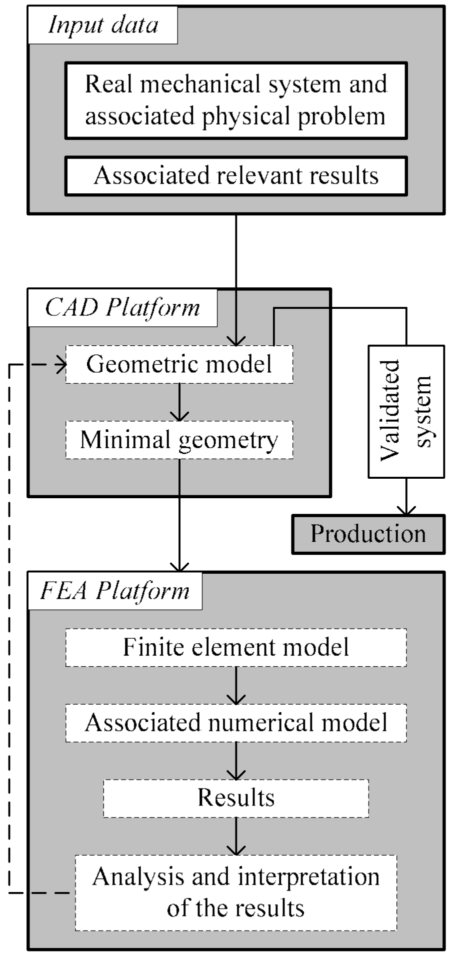

Fig. 2 – The precursory stages of the production of a mechanical system

Nature and relevance of the results obtained from solving numerical model associated with finite element model. Compliance with that principle implies that geometric modelling to be correlated with the nature and relevance of results used to examine the behaviour of the studied mechanical system.

Consider, for example, a mechanical system consisting of structural elements which can be assimilated with bars connected on articulated or rigid nodes. To study the behaviour of such a system under a loading system, it is more productive to use the results such as bending moments, torsion moments, axial forces, shear forces and the stresses due to these sectional efforts. For this reason, in such a case it is advisable simplified geometry modelling. The minimal geometry in this case is represented by the wire frame associated with the studied structure based on which on FEA platform will be generated a finite element model composed by one-dimensional finite elements (beam or rod).

Influence level of geometric perturbations on expected results. It correlates geometric modelling with increasing gradient of the targeted results. Thus, for example, when the influence on expected results induced by the geometric perturbations (holes, channels, fillet or chamfer with small dimensions, etc) are not significant, (where these, on the basis of Saint Venant principle are not in the vicinity of the areas where constraint conditions or applied loads are imposed) it is advisable to generate a simplified geometry. Consequently, in such case, minimal geometry will not include these geometric perturbations.

Existence of symmetries. It is necessary to correlates geometric modelling of the mechanical system under study with possible symmetries that it has: geometric symmetry, symmetry of material, symmetry of loading and constraining. If the previously mentioned symmetries exist simultaneously, minimal geometry consists of that part of the full geometry which is delimited by the axis and / or plane of symmetry. Existence and using model symmetries lead to finite element models that require fewer resources and also provide results with a good accuracy. However, the existence of symmetries does not imply a priori their use. In this context it is important to underline that also it is necessary a correlation with actual physical nature of the problem. Thus, for example, is not allowed to use symmetries in problems that involve modal analysis or in dynamics problems.

Modelling capabilities implemented on FEA platform. Current FEA platforms offer to user a wide variety of tools that allow simplified modelling of mechanical system parts without significant loss of accuracy of results. In this context mention the wizard type instruments that allow a simplified modelling of bolt connections and capability to determine the gravity centre of a plane area defined by the user.

Referring to bolt connections, it is important to note that depending on the level of detail needed in the area of the connection; these types of connections can be treated in one of the following manners. (The FEA model is usually an idealization of the real world, so details in the fastener may not be necessary.)

- Ignore the fastener itself and assume the parts are bonded together in the area of the connection. The loads are transmitted between the parts through a "full-strength" connection. Regions of the model remote from the connection will give accurate results. The results at the fasteners should not be used (except for providing results to use in a hand calculation of the fastener).
- Model the fastener as a beam or truss element along the centerline of the holes in the parts. The fastener is bonded on each end to the nodes of the hole, so slippage is assumed to not occur in the analysis. The results are reasonable in all areas of the model, given the approximation of the fastener to part interaction.
- Model the fastener with brick elements. The contact between the fastener and connected parts can either be bonded (no slippage), or contact between the parts can be included. These results are the most realistic that can be obtained at the expense of a much more complex model and longer runtime.

Next, referring to capability to determine the gravity centre of a plane area defined by the user it is important to note that this capability can lead to a major simplifying of geometry, as suggested in the example shown in Fig. 3.

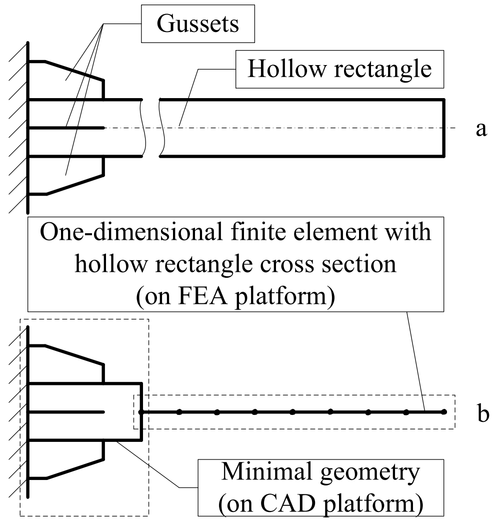

Fig. 3 – Simplifying geometry by using gravity centre of cross section
a – full geometry; b – minimal geometry stages of the production of a mechanical system

As suggested in Fig. 3, in cases similar to that shown in the figure, it is recommended that minimal geometry to contain only the part of full geometry that cannot be schematized as one-dimensional solid. On FEA platform, after determining the centre of gravity of the cross section of the beam end, for modelling the rest of the system will be used one-dimensional finite elements, not 3D finite elements (tetrahedral or hexahedral). Thus, finite element model is a mixed model because it contains one-dimensional and 3d finite elements (for minimal geometry, which is a 3d geometry). In this way is generated a small model with finite elements that provides results both accurate and relevant.

3. CASE STUDY

The case study refers to a connection with preloaded high strength bolts.

The connection fits into C category in accordance with valid actual standards, [2], and is designed to transmit a static axial effort between two elements made of steel flat strip. The used material is steel S 235, according to EN 10025-2.

The technical solution adopted uses two butt straps arranged on the sides of the elements loaded by tension (see figure 4 where are represented the characteristic dimensions of the connection). The butt straps are made of steel flat strip S235, according to EN 10025-2.

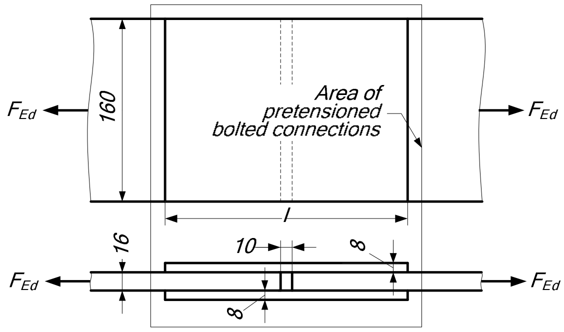

Fig. 4 – Adopted technical solution for studied connection

In order to achieve the connection the HV system is adopted, using 12 high-strength pretensioned bolts arranged in three rows and four columns, as shown in Fig. 5. The connection has a symmetry plane which is perpendicular to the axial transmitted effort.

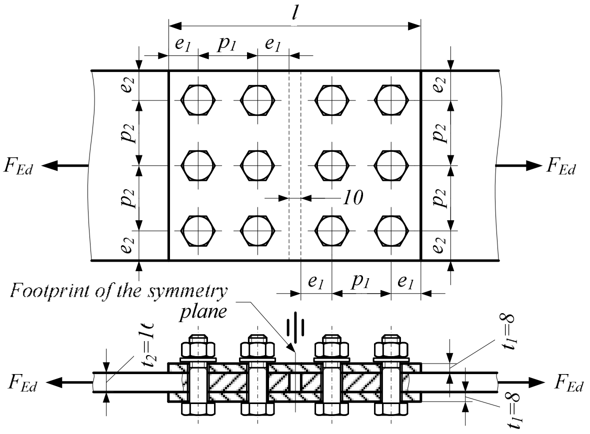

Fig. 5 – Adopted connection

Because of the symmetry, is considered in the study only one side of the connection resulting from the intersection of the plane of symmetry.

Referring to figure 5 we mention that the characteristic dimensions of the connection were adopted in accordance with the regulations contained in [2].

In Fig. 6 is shown the full geometry of connection and in Fig. 7 is shown the associated minimal geometry which was imported on FEA platform in order to finite element modelling (Fig.8).

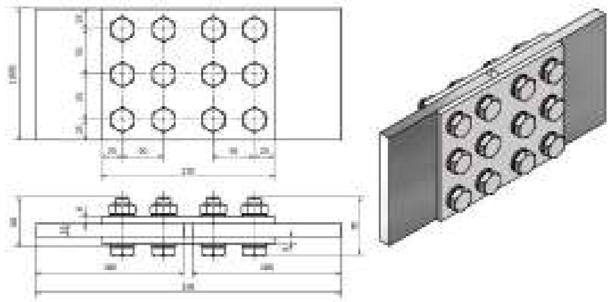

Fig. 6 – Full geometry of connection

In the minimal geometry, butt strap halves were generated as reunions of two identical parts having contact in plane of symmetry containing the axis of the central holes. The purpose of this approach is to ensure presence of finite element nodes in axial plane of the connection, nodes which are needed to highlight the relevant results in this plan. In Fig. 8 is shown finite element model associated with minimal geometry.

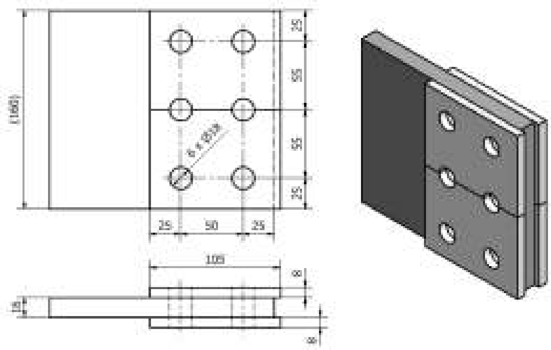

Fig. 7 – Minimal geometry associated to the studied connection

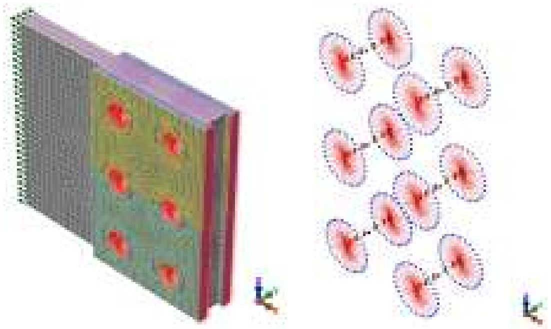

Fig. 8 – Finite element model of the connection

4. CONCLUSION

The engineer can approach modelling geometry of a mechanical system in two ways:

- Geometry modelling in order to obtain technical documentation;
- Geometry modelling in order to obtain finite element, model which, usually, requires minimal geometry.

Although we can imagine rules and algorithms that enable automatic obtaining of the minimal geometry, however, the user experience and analysis objectives have an essential importance. Using minimal geometry, generated based on the above principles will lead to optimized models with finite elements, models which require fewer calculus system resources and also leading to results with a good accuracy.

REFERENCES

[1] Dumitrache, P., (2014). *Considerations on The Simplified Numerical Simulation of The Bus Lateral Rollover* - The Annals of „Dunărea de Jos” University of Galaţi, 2013, Issue 2, Fascicle XIV Mechanical Engineering, pag. 43-50, ISSN 1224-5615.

[2] *** *EN 1993-1-8: Eurocode 3: Design of steel structures*

[3] *** *Autodesk Mechanical Simulation – User manual*

Author:

Eng. **Petru DUMITRACHE**, Ph.D., Assoc. Professor, Engineering and Management Sciences Department, “DUNĂREA DE JOS” University of Galaţi, Engineering Faculty of Brăila, E-mail: dumitrache.petru@gmail.com