TCT
CONFERENCE 99
EUROPE

Simulation Driven Design Process: A Case of Paradigm Shift

Paul M. Kurowski

Dr. Paul Kurowski

ACOM Consulting
pkurowski@acomdesign.com

Dr. Paul Kurowski has a M.Sc. and a Ph.D. in Applied Mechanics from Warsaw Technical University. He has completed post-doctoral work at Warsaw Technical University, Kyoto University, and the University of Western Ontario. He is President and Owner of ACOM Consulting, a consulting firm specialising in Computer-Aided Engineering (CAD, FEA, etc), product development (automotive, medical, etc.) Dr. Kurowski is also a professor (adjunct) in the Department of Mechanical Engineering at the University of Western Ontario. He is a member of the Society of Automotive Engineers and the Association of Professional Engineers of Ontario. He is a Certified Engineering Provider for the Parametric Technology Corporation in the field of Functional Simulation.

1. Introduction

Simulation tools of Computer Aided Engineering (CAE) such as Computer Aided Design (CAD), Finite Element Analysis (FEA), Computational Fluid Dynamics (CFD), Dynamic Analysis etc. are now common in engineering offices. Engineers and designers use them to shorten the design process by replacing physical prototypes with simulation models. That later translates into competitive advantage and increased profits. However, since the same tools are also available to the competition, the competitive advantage can not be gained just by implementing simulation tools. The real advantage can be gained only by proper implementation of the most effective and powerful simulation tools.

This paper will limit the scope of discussion to the role and place of Finite Element Analysis (FEA) simulation in the product design process and focus on two topics:

- the way simulation should be implemented in the design process
- required software features to make that implementation successful

2. Implementation of FEA in the Product Pesign Process

Typically, a new design is developed in CAD and its geometry includes all details required for manufacturing — we will call it CAD specific geometry. The design is then submitted to FEA simulation, which requires conversion of CAD specific geometry into simulation specific FEA geometry. Next, results of FEA are used to modify the design. Design modifications, in turn, need to be verified by FEA etc. During this iterative process we observe frequent reciprocating between CAD specific geometry and simulation specific FEA geometry while the design is gradually refined.

CAD geometry must contain all the information required for manufacturing which results in high levels of detail. Yet, details which are not structurally important should be removed or suppressed when creating FEA simulation specific geometry. Simulation specific geometry may also need to be idealised by using zero thickness wall representation for shell element meshing or introducing a stick model for beam element meshing. Furthermore, simulation specific geometry may take advantage of symmetry, anti-symmetry, or cyclic symmetry and include only a portion of the analysed structure. Before we discuss implications of differences between CAD and FEA geometries on FEA implementation in the design process, we will illustrate those differences with a few real life examples.

2.1 Booster Arms

A CAD model (see figure 1) includes, as it should, all of the details required for manufacturing. However, it would be practically impossible to use it "as is" for FEA simulation. Even though CAD produced solids could be meshed with solid finite elements, the resulting solid model would require a very large number of elements making it very expensive to use. Considering practical time and hardware limitations, the solid model is out of the question. Instead, the geometry must be converted into one that is suitable for shell element meshing (see figure 2).

2.2 Bracket

The CAD model (see figure 3) offers all of the details important for manufacturing. The simplicity of bracket geometry seems to make it a good candidate for meshing "as is" with solid elements. However, upon close inspection we realise that the geometry still needs some work before it can be used for a FEA model. Small fillets, which have no structural importance and would only unnecessarily complicate the model, need to be removed (see figure 4).

2.3 Cylinders in Contact

Our objective here is to find contact stresses between two cylinders. As figure 5 shows, CAD produced geometry could not be any simpler than that. Still, it should not be meshed without some preparation. Contact stress analysis requires that small enough elements are placed in the contact zone, so contact pressure could be modeled properly. Therefore, volumes representing each cylinder may need to be split into two or more so mesh size in the contact area could be easily controlled. Portions distant from the contact zone can be removed to simplify the model. Additionally, we may also take advantage of symmetry, finally producing simulation specific geometry as shown in figure 6.

Considering the differences between CAD and FEA geometry, we face the task of alternating between those two during the concurrent CAD-FEA design process. As shown in figure 7a, CAD and FEA are done concurrently to shorten the process: CAD design is used to define FEA models, and FEA results are used to modify CAD models. This is an iterative process, the essence of concurrent product development. However, each reciprocation consumes time and runs the risk of errors. In addition, and maybe most important of all, new FEA results may negate CAD work that has progressed in the mean time. To avoid frequent "FEA round trips" we propose to modify the concurrent CAD - FEA process where design starts in the CAD domain into an FEA simulation driven product design process. The FEA simulation driven product development process will start in the FEA domain and stay there throughout all design iterations (figure 7b). On completion, the idealised FEA specific geometry will be converted into CAD specific geometry. Hopefully, that will only need to be done once.

Notice that while suggesting the process shown in figure 7b, we do not propose to use FEA software for conceptual design. While this is conceivable, engineers usually find CAD better equipped to handle geometry, both CAD specific and FEA simulation specific. Therefore, a concurrent as well as a simulation driven design process will benefit from close integration between CAD and FEA as discussed in the next chapter.

3. Features of CAD and FEA Software for Effective Implementation of Simulation in the Design Process

Implementation of FEA in the design process requires effective exchange of information between CAD and FEA, which are two different pieces of software. We will now review which features of CAD and FEA software and which features of the interface between the two, will benefit the design process, either concurrent or simulation driven.

3.1 CAD System Features

A CAD system should:

- create 100% of complex geometry: both CAD specific and FEA simulation specific
- offer quick alternations between those two geometries while keeping them linked
- optimise geometry prior to FEA simulation (behavioural modelling)
- offer tools for communicating design intent to the rest of organisation
- offer close integration with FEA.

The above CAD system requirements call for parametric, associative, feature driven formulations to make quick alternations between CAD and FEA geometries possible. A highly desirable feature of a CAD system contributing to successful implementation of simulation, is the ability to optimise geometry, prior to FEA simulation. This feature, called behavioural modelling, allows engineers to perform FEA simulation on already optimised geometric layout. Finally, a successful design process is aided by simple to use kinematic and animation tools allowing other areas of the organisation to see design intent earlier in the design process

3.2 FEA System Features

A FEA system should:

- mesh relevant CAD produced geometry automatically (user's intervention should not be required)
- map finite elements precisely to geometry
- offer discretisation error assessment
- offer optimisation and sensitivity studies capabilities
- offer close integration with CAD.

The FEA system requirements call for associative p-element FEA software. We will now discuss why p-element formulation is preferable for effective implementation of FEA in the design process.

3.2.1 Automeshing with no user intervention and mapping to geometry

Automeshing of CAD produced geometry is purely geometric exercise, the unsupervised automesher simply fills up the available volume (or surface) with elements. It does not know anything about the expected stress field so it simply selects element size according to some characteristic geometric features such as wall thickness, placing one layer of elements across the wall. The quality of resultant mesh very much depends on whether h or p elements are used.

Using h elements, one layer of elements across the wall is a dangerous proposition. Considering that h elements are of the 1st or 2nd order, one layer of elements can not possibly model e.g. bending so the model is either meaningless or dangerous depending on the importance of our analysis. To make it even worse, an h element automesher often finds it impossible to create elements of correct shapes as h elements are quite sensitive to degeneration. As a result, we often see h element meshes with a large number of degenerated elements.

A p element automesher will also place one layer of elements across a wall in bending. However, due to element adaptive order (e.g. up to 9th order in Pro/MECHANICA) one layer of elements across a wall is perfectly adequate. Also, p elements are less sensitive to degeneration; what looks like a really bad h element is still an acceptable p element so degeneration is less of an issue using p elements.

Regarding mapping, h elements approximate geometry with straight edges and faces, p elements can assume curved edges and faces so they map precisely to geometry. The differences between the results of h and p element meshing of a solid geometry is shown in figure 8. While it is certainly possible to create a correct h element model, that would require user intervention to define element size, number of nodes along each line etc. In contrast, p elements mesh correctly without any need for user input or judgement.

3.2.2 Discretization Error Analysis

All FEA results are always produced with error and that error must be known before we can base design decision on FEA results. The objective of FEA used to support the development process can be stated as: "Compute the data of interest and prove that the associated error is small". p software meets that objective offering automatic, "hands-off" discretisation error analysis performed by iterative element order upgrade. In contrast, discretisation error analysis with h-elements is a tedious process calling for several different models with an increasingly refined mesh. In practice that is seldom done and design decisions are often made based on data of interests with unknown errors!

3.2.3 Optimisation and Sensitivity Studies Capabilities

Using associative, p element formulation, optimisation and sensitivity study capabilities are easy to implement and to use. Due to the "high resistance" of p-elements to degeneration, the same mesh can usually be used throughout the entire optimisation process. If re-meshing becomes necessary, it can be done without user intervention.

3.3 Features of the Interface between CAD - FEA

Users should have a choice of different levels of integration between CAD and FEA:

- Integrated mode — all analysis functions are done from inside the CAD.
- Linked mode — geometry is controlled by CAD, the rest by FEA.
- Independent mode — all functions, including geometry, controlled by FEA.

4. Conclusions

The success of FEA simulation depends on the way it has been implemented in the design process, the type of FEA software used, and user skills. Here, we addressed the first two issues. We proposed to start and iterate our design in the FEA specific geometry, avoiding unnecessary interfacing between CAD and FEA. This will also avoid CAD work that might have been done while FEA analysis was in progress, only to become obsolete in view of new FEA results. The fact that design is commonly started in CAD geometry appears to be a legacy from the days when simulation tools were not available. The simulation driven design process does not minimise the importance of effective CAD-FEA interfacing. It is still preferable to use CAD for both CAD specific and simulation specific geometry and a tight integration between CAD and FEA is desirable.

For FEA simulation, we propose to use p-element software (e.g. Pro/MECHANICA) which offers the following major advantages over traditional h-element software:

- Reliable meshing of geometry of any complexity with little (if any) need for users judgement.
- Precise mapping to geometry due to p-element.
- Iterative solvers offering automatic discretization error analysis.
- Easy implementation of optimisation and sensitivity studies capabilities.

What we propose requires a paradigm shift in design engineering culture: a new approach to CAD design and FEA simulation putting simulation first and using software well suited for supporting the design process.

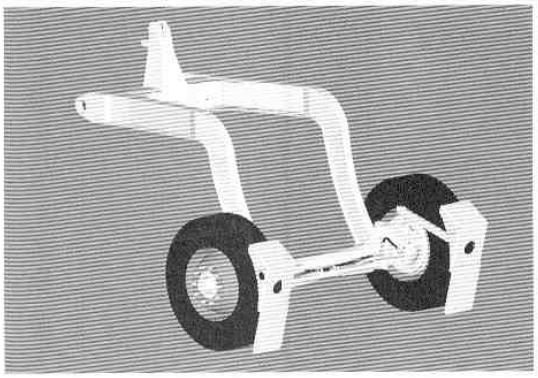

Figure 1. CAD model of booster arms for concrete mixer truck.

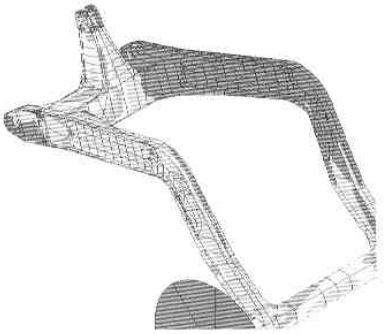

Figure 2. Results of FEA of booster arms.

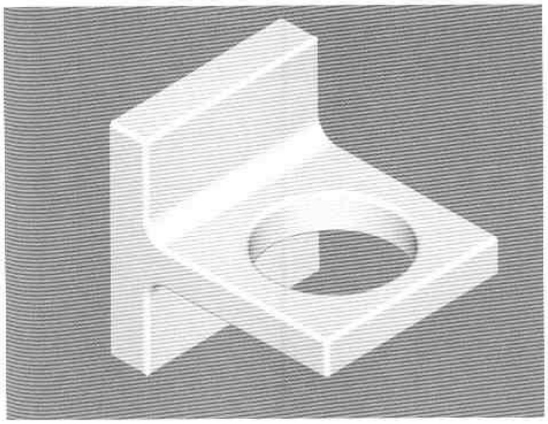

Figure 3. CAD model of cantilever hollow bracket.

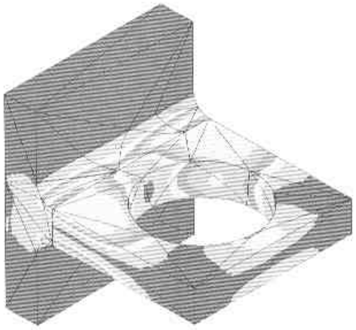

Figure 4. Results of FEA of cantilever hollow bracket.

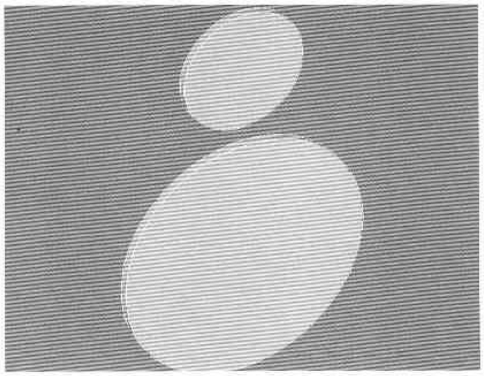

Figure 5. CAD model of two short cylinders in contact.

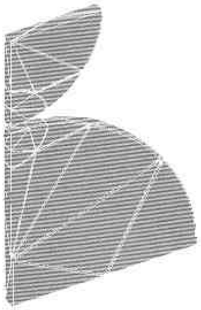

Figure 6. Finite element model for contact stress analysis.

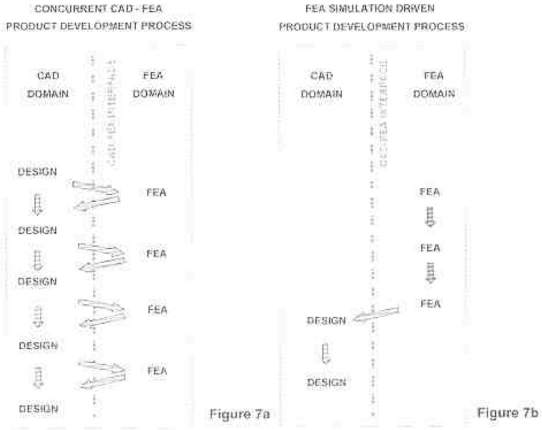

Figure 7a

Figure 7b

Figure 7. Concurrent and simulation driven design process.

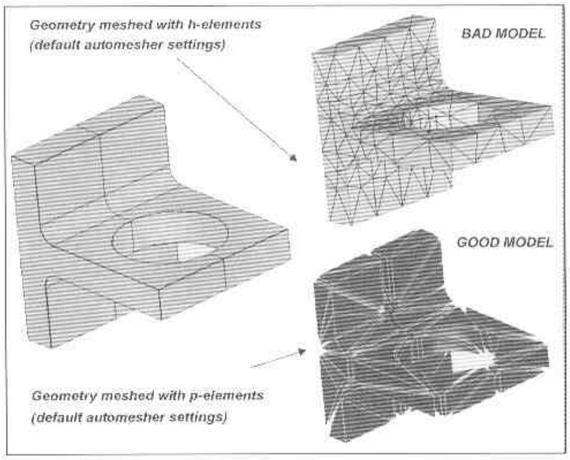

Figure 8. Results of automeshing with solid elements using h elements and p elements.