



LUND UNIVERSITY

On the Development of a Modeling Tool for Collaborative Finite Element Analysis

Burman, Åke; Eriksson, Martin

2002

[Link to publication](#)

Citation for published version (APA):

Burman, Å., & Eriksson, M. (2002). *On the Development of a Modeling Tool for Collaborative Finite Element Analysis*. Paper presented at 2002 ANSYS 10th International Conference and Exhibition.

Total number of authors:

2

General rights

Unless other specific re-use rights are stated the following general rights apply:

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal

Read more about Creative commons licenses: <https://creativecommons.org/licenses/>

Take down policy

If you believe that this document breaches copyright please contact us providing details, and we will remove access to the work immediately and investigate your claim.

LUND UNIVERSITY

PO Box 117
221 00 Lund
+46 46-222 00 00

Development of a Modeling Tool for Collaborative Finite Element Analysis

Åke Burman and Martin Eriksson

Division of Machine Design, Department of Design Sciences, Lund Institute of Technology at Lund University, Sweden

ABSTRACT

In today's competitive society the difference between winning and disappearing might very well depend on who is first to release a new product or an update of an existing product to the market. The impact that this situation has on the design analysis department in a company is tremendous. It is a known fact, to everyone working with design analysis, that the demands on analyses from the product development and design department in a company are constantly increasing with respect to complexity, quality and quantity. One of the most obvious reasons for this is most likely that the engineering designers of today have tools at their disposal to generate new geometries and variants very fast and accurately. The introduction of fully parameterized, two way associative, 3D CAD programs has revolutionized the way engineering designers work today compared to the situation just a decade or two ago.

The implemented Internet/intranet based FE-modeling tools will enhance the possibility for design engineers to rapidly and efficiently generate new variants of a given design concept. The tools are implemented with the aim to give engineering designers the possibility to add/replace predefined parts and/or components to an overall FE structure.

Introduction

At present new concepts such as “Collaborative Engineering” are being introduced to enhance the productivity of a design team even though the design team is split up and working on the same task from different locations (sites). This development has been possible due to the improved communication facilities that are tightly linked to the use of computers and high-speed networks.

Unfortunately, the development of tools for performing analyses, e.g. Finite Element Analysis (FEA), has not occurred at the same pace as for the CAD tools. The great improvement of FE programs that has taken place during the last decade or so should of course be recognized. A number of new mesh methods, solvers, material models, optimization tools, element types and so on have made the analyst's life a lot easier. However, today it is fairly simple and straightforward to mesh rather complex geometries and the actual solution of a moderately large FE model, let's say with approximately 100,000 DOF; this is more or less automatically performed in just a few minutes on a moderately equipped PC.

The main drawback of today's FE -programs is their poor ability to rapidly and efficiently generate new models (geometries). Even changing an existing geometry slightly might be a challenge to an experienced FE analyst. As a consequence, the main resource consumption is devoted to modeling in the form of man-hours. One might argue that it is possible to transfer geometry from a CAD program to a FE program via e.g. IGES. In theory this is an interesting approach, but in practice very few geometries are such that they can be successfully transferred without spending hours on repairing and fixing the imported geometry.

To improve the situation for the analyst, the modeling activity has to be improved on significantly. The authors experiences from a substantial number of analysis projects focused on FE analysis is that most models developed are variants or minor revisions of already existing models. In addition, a large portion of most models developed consists of predefined, and prefabricated, standardized machine elements. Another scenario is that company-specific standard components or subsystems, of different sizes or capacities in some respect, are frequently being used.

Objective

Based on the above-mentioned observations and experiences, a development project with the objective to develop a modeling tool to enhance and improve the situation with respect to modeling is in progress. The objective of the project can be summarized as follows:

To create an Internet/intranet based modeling tool to enhance the creation of variants of a given design concept. With this tool it should be possible to add or replace predefined parts and/or components of an overall, well-defined, structure. Using the tool should be easy and thus not require any analysis or modeling expertise.

Approach / Implementation

The implementation presented in this work is a Client/Server WWW application that can reside within the Internet or on the local intranet. The implementation can be divided up into two major parts, the user interface, see Figure 1, where the User interacts with the implementation, and the server backbone. The user interacts with the web server that collects the input data and stores it on the server hard drive for further use. Further, when the user requests an analysis, the web server interacts with the FE preprocessing application and the FE analysis file is created. The analyses are then performed and the web server collects the resulting data, which are then presented under the results directory in the project management tree to the left in Figure 1.

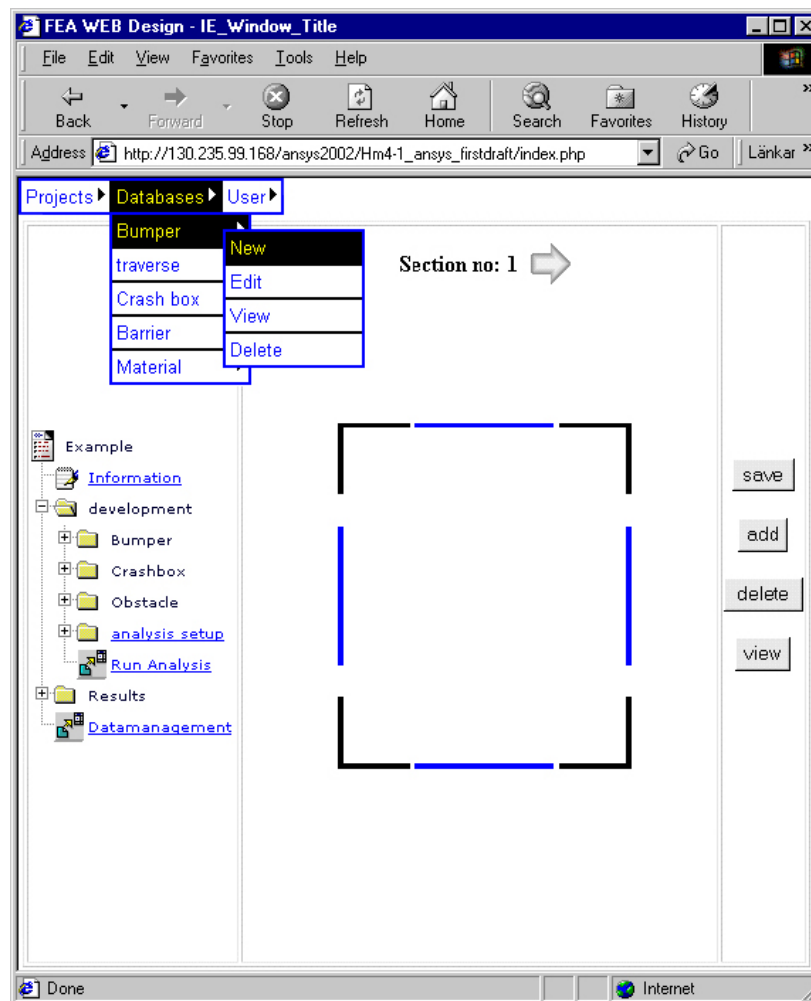


Figure 1 - Principle layout of the implementation

The user interface is divided up into two separate functions. At the top of the HTML page there are three major choices to be made:

- Projects; part where projects can be created, edited and also deleted. The project is the highest system level in the implementation and it contains input data as well as result data.
- Databases; contains information on different sub-systems along different components that could be utilized in the projects.
- User; contains information on the current logged-in user. The user can manage his own projects along with the personal information stored for the user.

The second part is where the user enters the data for the current project or the work on a sub-system or component definition.

Figure 2. Displays an example of an automotive bumper analysis project where a project named Example is created, as can be seen at the top of the left table cell. The project consists of a bumper with 6 defined cross sections and a crash box along with a number of applied analysis options.

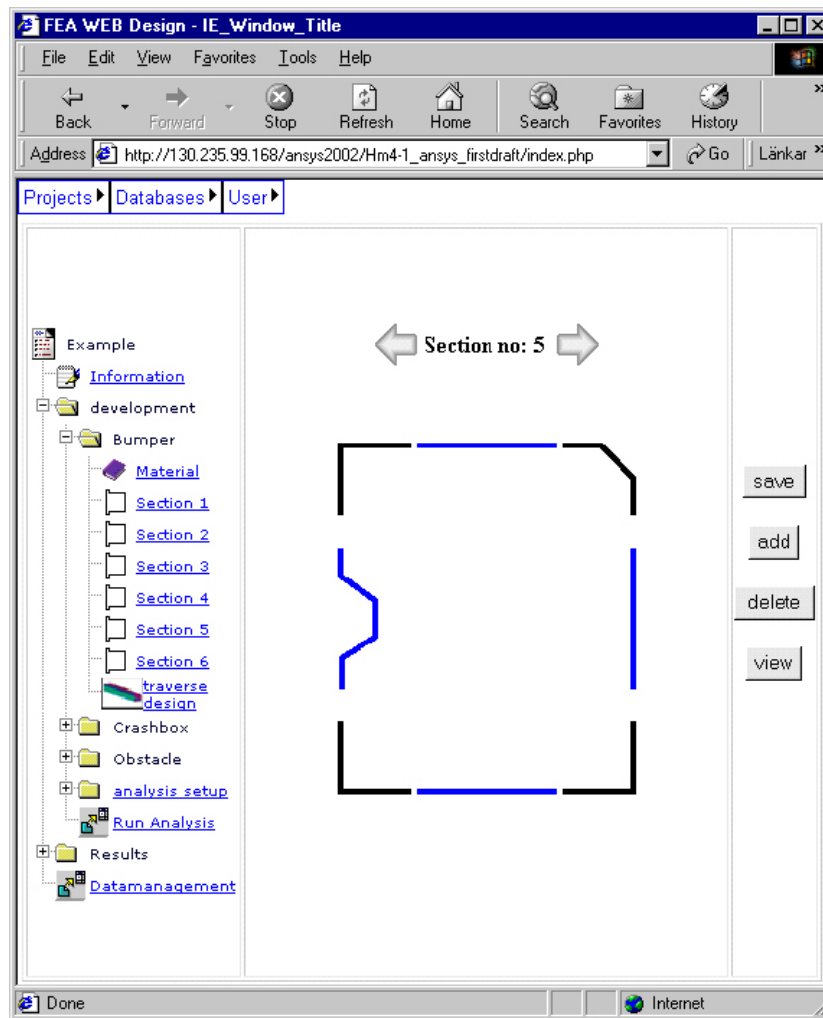


Figure 2 - Principle sketch of automotive bumper analysis project user interface.

In the middle of Figure 2 the cross section number 5 is highlighted. The user has the ability to alter the basic shape of the cross section by clicking on one of the four corner components or the four mid side parts, as can be seen in Figure 3a and Figure 3b.



Figure 3 - Principle layout of the corner and mid side sections

The server calculates when all the basic shapes of all cross sections are outlined. The number of points building up each cross section and the geometry of the bumper can now be defined and positioned by the HTML page displayed in Figure 4. For each one of the points, coordinates are defined in either a global coordinate system or in the sections' local coordinate system. The section is also positioned and arranged with respect to the other cross sections. Further, the thicknesses of each part of each cross-section are defined to the right in Figure 4.

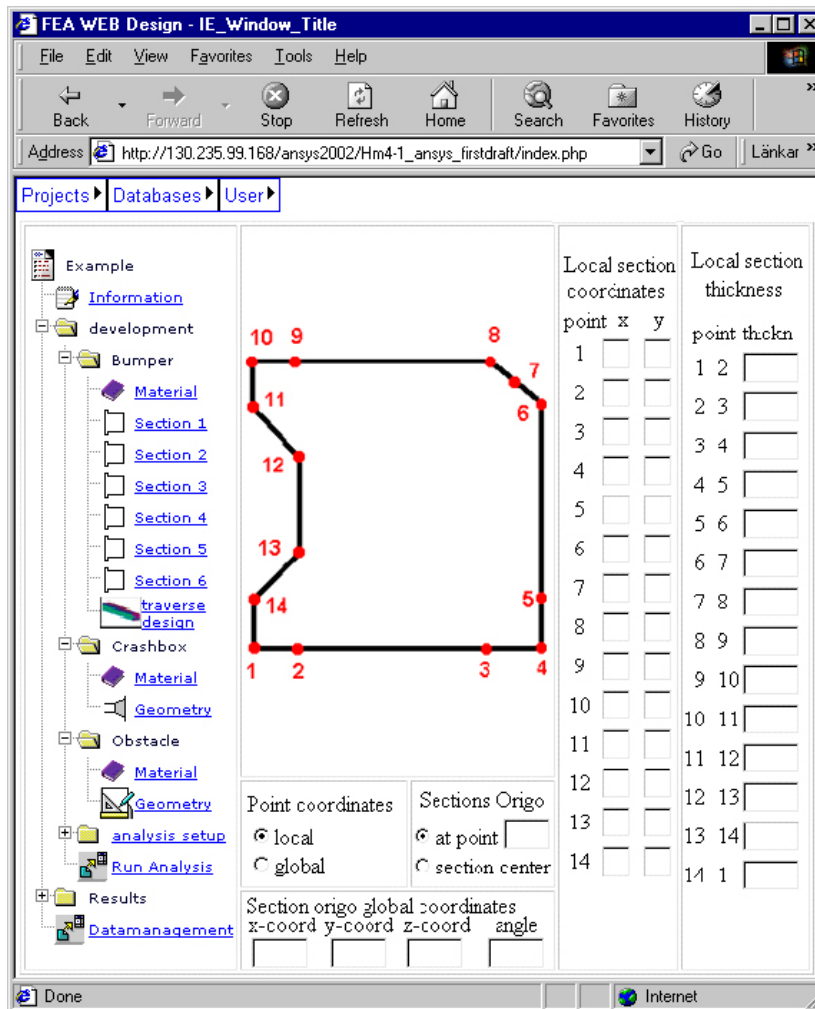


Figure 4 - Cross section definitions of the bumper cross sections

An analysis Example

The example chosen to illustrate the implementation is a conceptual development project of a bumper in a transportation device. The bumper is a critical component in the device since the operator of the device is located close to the component. To assess the behavior, different types of load cases are tested on the bumper. The load case shown in this example is a barrier impact (crash) analysis. The criterion for the conceptual bumper design is that the component should result in low contact interface force with highest possible energy absorption. To facilitate the flexibility in the application, two different designs have been created and evaluated. The design layouts of the bumpers are shown in Figure 5. In the top row of the figure each one of the six sections is displayed, and in the middle row the basic shape of the bumper is shown. Connecting each corresponding point in the cross sections by spline curves creates the bumper. The area plots of the bumpers in the bottom row highlight the bumper surfaces used in the subsequent analyses.

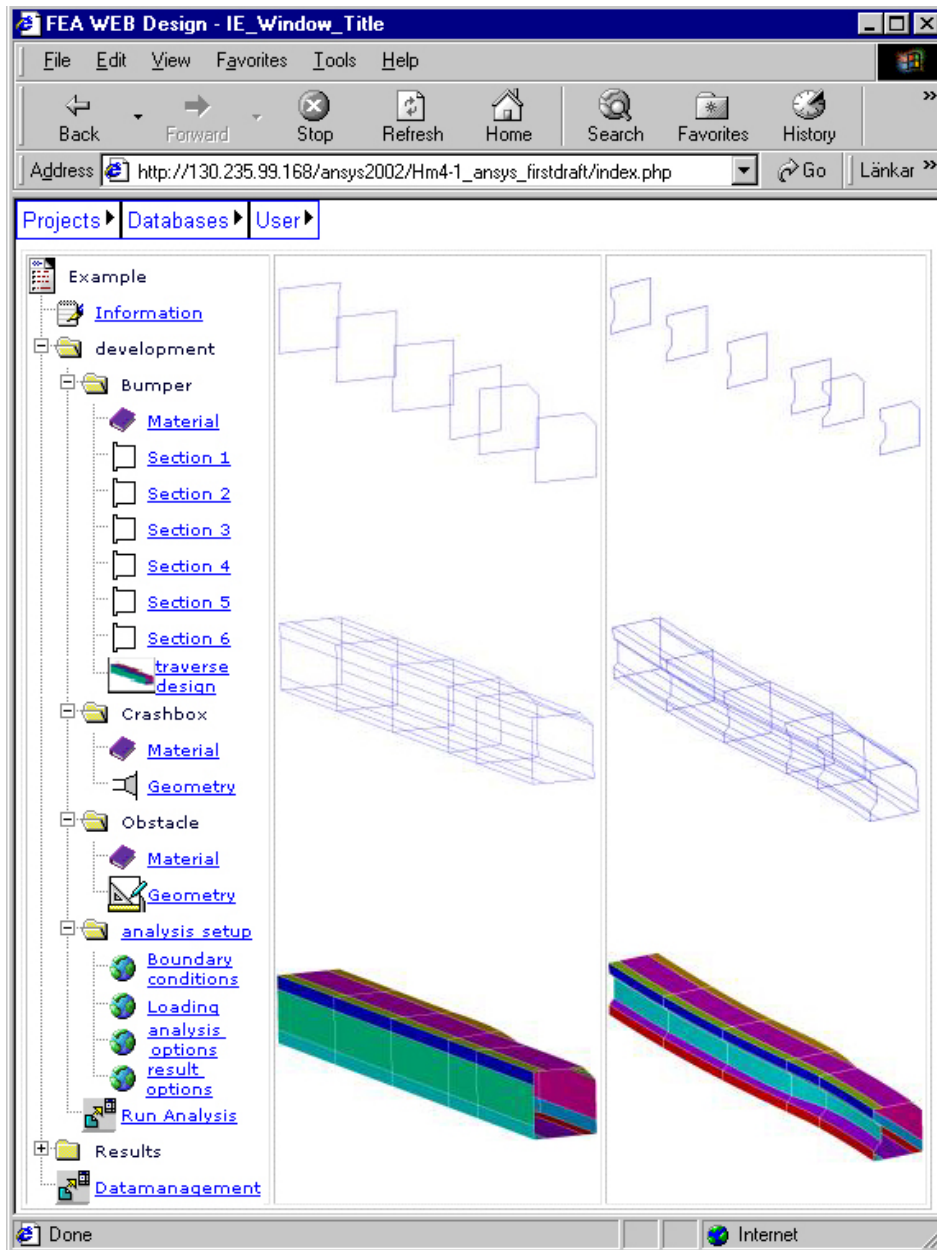


Figure 5 - Principle layout of the example Bumper 1 and Bumper 2

The bumper is modeled with half symmetry such that in each bumper analysis twelve cross sections are used to build the complete bumper, as can be seen in Figure 6 along with the crash box and the barrier that the bumpers are crashed into. The analyses are performed by giving the crash box and the bumper an initial velocity towards the barrier that is constrained in all directions throughout the whole analyses.

The results chosen to be extracted from the analyses are the Equivalent Von Mises stress that is displayed in Figure 6.

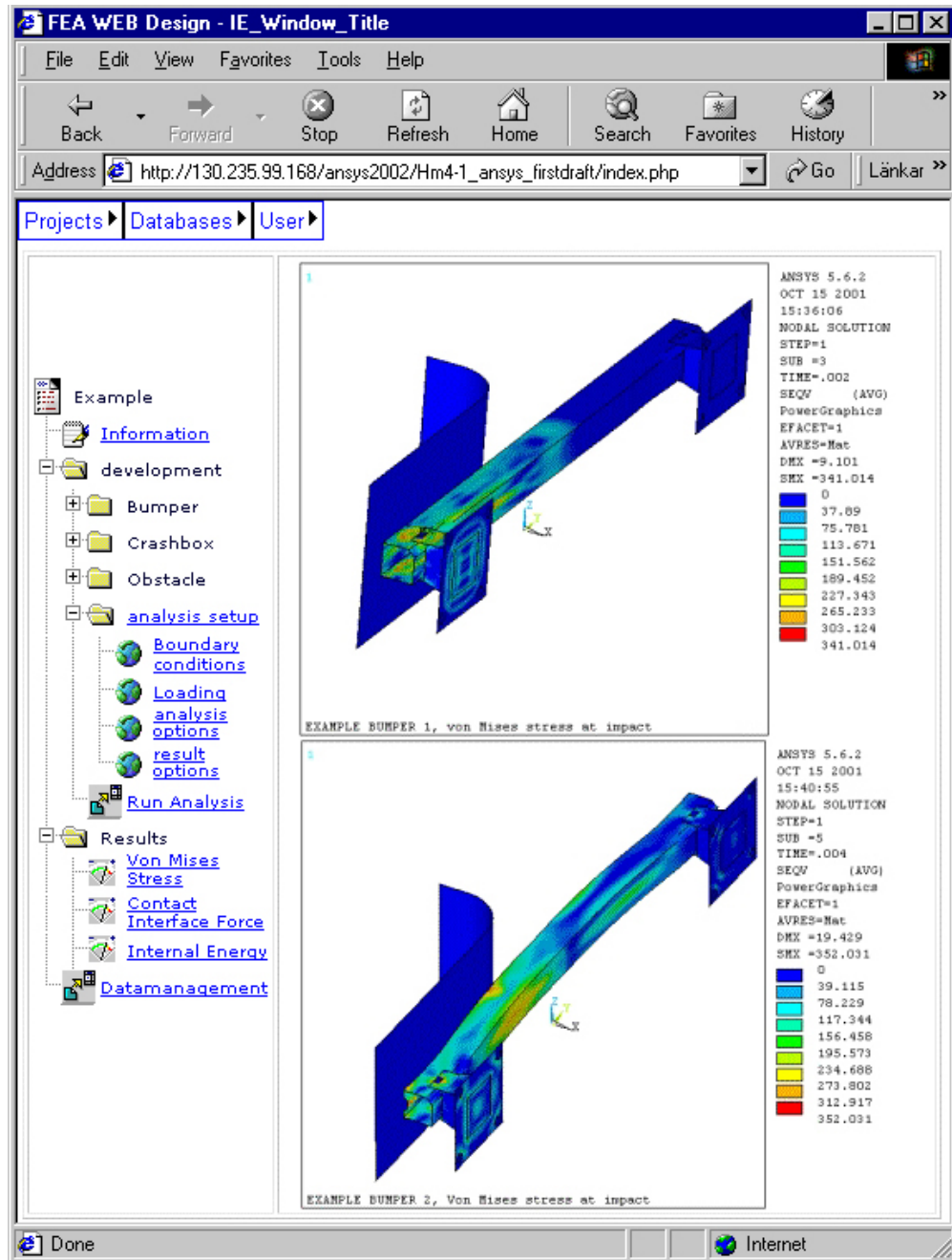


Figure 6 - Von Mises stress of the bumpers

The two criterion objectives are also extracted, and the contact reaction force is displayed in Figure 7 and the internal energy absorption is displayed in Figure 8.

As can be seen in the results, the second bumper design better fulfills the criterion established for the analyses and thus is the most promising design concept to evaluate further.

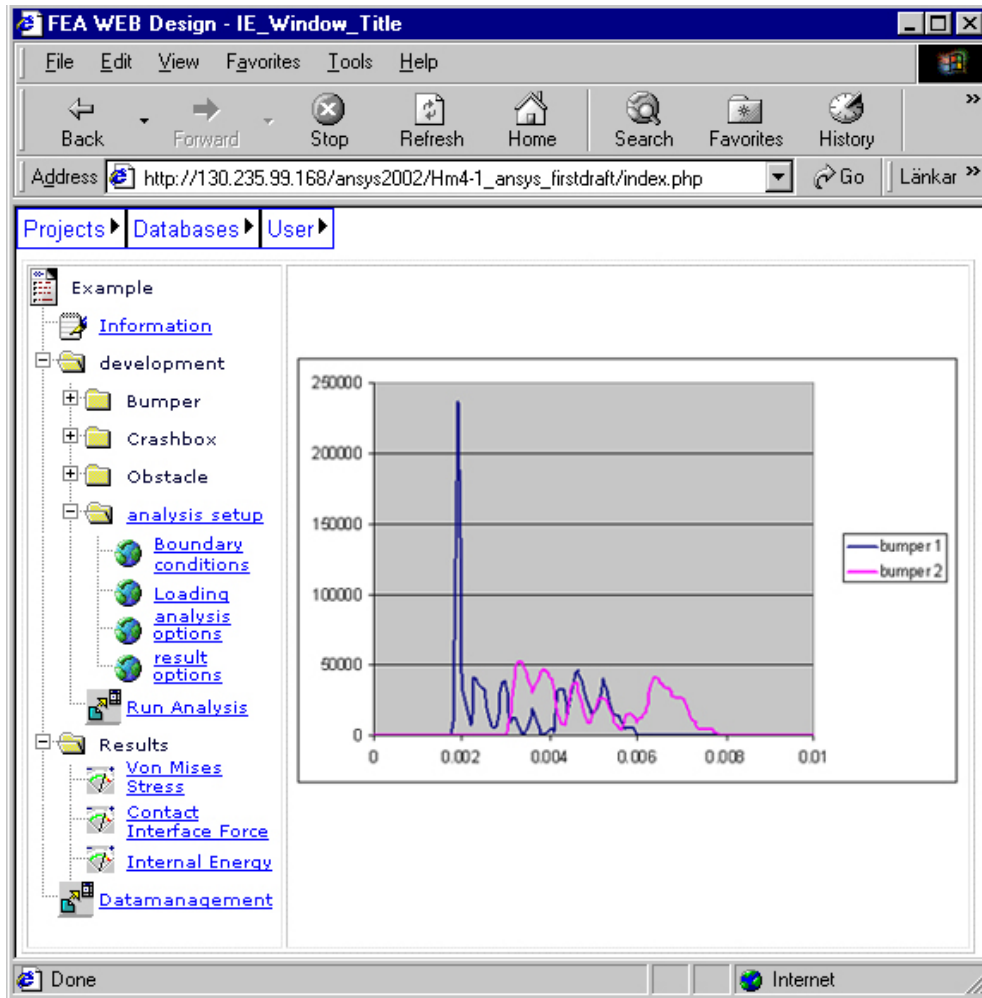


Figure 7 - Contact interface force of the example Bumpers

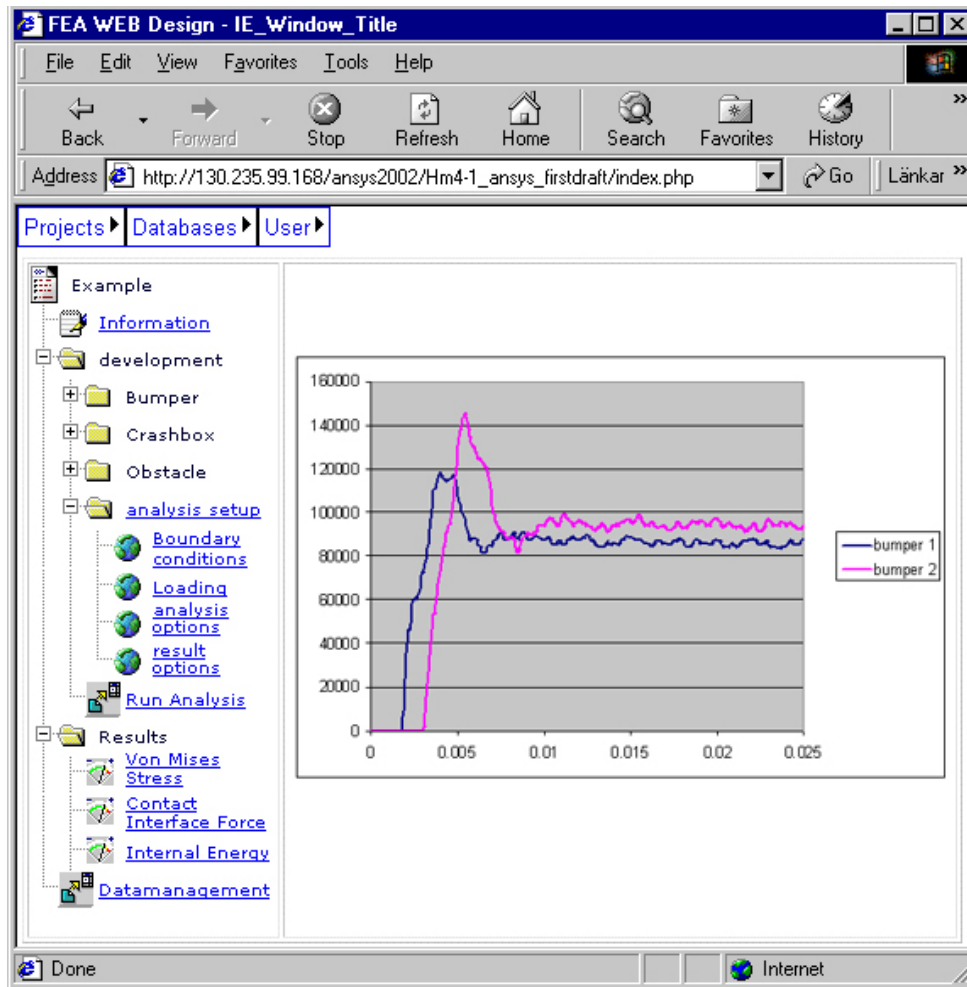


Figure 8 - Absorbed internal energy in the example Bumpers

Further Developments

To enhance the modeling tool even further, a number of improvements will be implemented in the near future. At present the following improvements are scheduled for implementation.

- Further enhancements on the Web based interface of the client/server approach.
- Integrated database management to create and retrieve standardized parts and subsystems. The database server can be located on an intranet (LAN) or internet (WAN).
- Automatic update of results to a database for easy review of previously performed analyses and solutions/configurations presented as response surfaces.
- Support for models of different levels of simplification for different stages of the design process.
- Result presentation in both listings and graphics. For easy post-processing, VRML is utilized for all graphics.
- Automatic generation of analysis report in HTML.
- Integrated Probabilistic Module for stochastic FE analyses.

Conclusion

Based on extensive use of the modeling tool as described, the authors have achieved significant timesavings in a vast number of projects. This is particularly true of projects where a large number of different geometries are to be investigated with respect to minor changes in the geometry. In comparison to utilizing the optimization module in ANSYS, one can conclude that the optimization module could probably arrive at the same, or possibly better, solution. With the use of a web-based interface the user does not have to be an ANSYS expert with deep knowledge of APDL, but simply an engineering designer with rudimentary FE knowledge. Also, to do the programming directly in APDL for each geometry to be investigated would be, although possible, very demanding in order to manage all parameters and model variations. Further, the use of direct programming in APDL cannot really be considered an alternative for this application since the possibilities for a general interface where different FE-specific (ANSYS, LSDYNA) input decks can be generated will be lost, and also, when the data management of the implementation are to be updated with a database system, a general approach is favorable.