# META AS A POST-SOLUTION CALCULATOR FOR USER DEFINED CROSS-SECTIONS PREVENTS FROM RE-SOLVING CRASH MODELS AND FACILITATES SUBSTRUCTURING FOR DURABILITY MODELS

**Stavros Kleidarias, George Kalaitzidis**
BETA CAE Systems S.A., Greece

KEYWORDS:

Cross Sections, Section Forces & Moments, Substructuring

ABSTRACT:

Throughout the design cycle of a structure using the finite element method, the calculation of section forces at selective regions proves to be an important aspect to accurately forecast the structure's behaviour and monitor the way forces are transmitted within it.
For crash analyses, forces are calculated only for cross sections predefined in the solver's input file, thus making a second run unavoidable in case section forces should had been requested also for other regions.
For static analyses, the resultant forces over cross sections are not available by the solver thus not providing full insight of the structure's behaviour. Moreover, in case of performing substructuring out of the solver, it should be possible to use these results readily as loads for the region to be substructured.
This paper presents a new tool in META for the calculation of section forces from existing results extracted from several implicit and explicit solvers. Forces and Moments on any cross section of a crash model can be calculated by the Section Forces tool of META, having very good correlation with those calculated by the respective solver.
Resultant forces can be graphically visualized in 3D display on the model and can be output in the respective format to be used in the Boundary Conditions' definition of the substructured region.

TECHNICAL PAPER

## 1. SECTIONS RESULS CALCULATIONS IN META

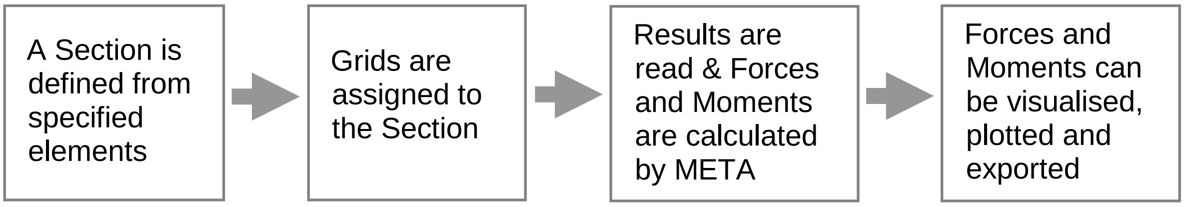
<u>General</u>

The Section Forces tool of META is a tool for graphically displaying and/or calculating grid point forces and moments from results file of the supported solvers: NASTRAN, ABAQUS, LS-DYNA and PAMCRASH.
The calculations are based on results available if certain output requests have been specified in the input deck of the finite element model, which depending on the solver include:

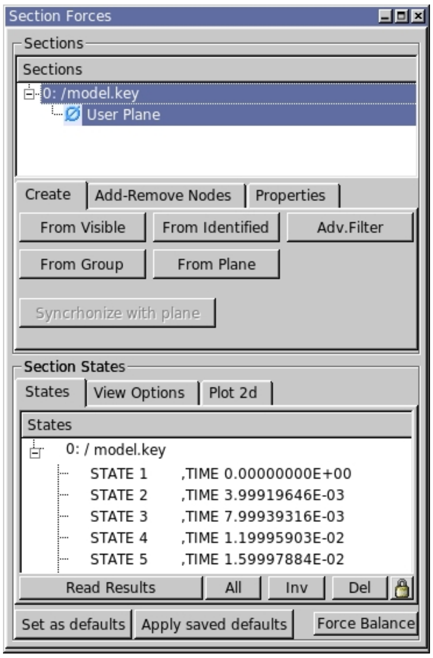
| | |
|---|---|
| **NASTRAN:** | - Grid Point Forces (GPFORCE output control command) either in op2 or punch file. |
| **ABAQUS:** | - All components of Point Loads and Concentrated Moments (related variable identifiers: CF, CM)<br>- All components of Reaction Forces and Moments (related variable identifiers: RF, RT, RM)<br>- Internal Forces at the nodes of elements (related variable identifier: NFORC)<br>- The above ABAQUS results should be available within the odb file as FIELD results.<br>- Contact Normal Force (related variable identifier: NFORC) |
| **LS-DYNA:** | - Element Forces in the d3plot file. |
| **PAMCRASH:** | - Element Forces in the DSY file. |

## Calculating Forces and Moments from user-defined Sections

The concept/workflow for calculating Forces and Moments for Sections in META is as follows:

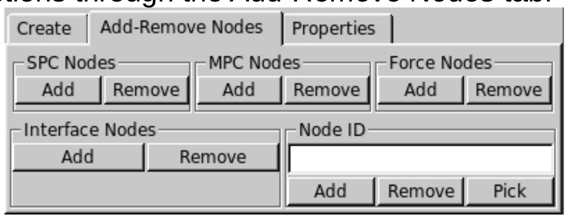



The Section Forces tool offers various options for the sections' creation:



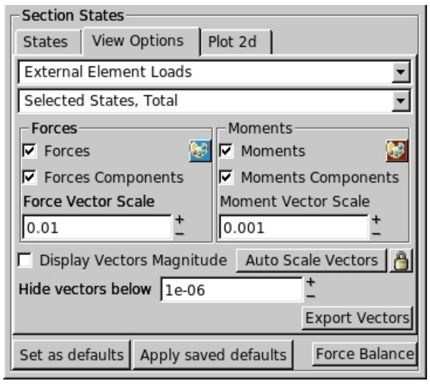

Sections can be created from *Visible* and *Identifed* elements, from *Groups* (including Node Sets defined in the input deck), from existing *Planes* (including Sections defined in the input deck) and by specifying filtering criteria in the *Advanced Filter* tool of META.

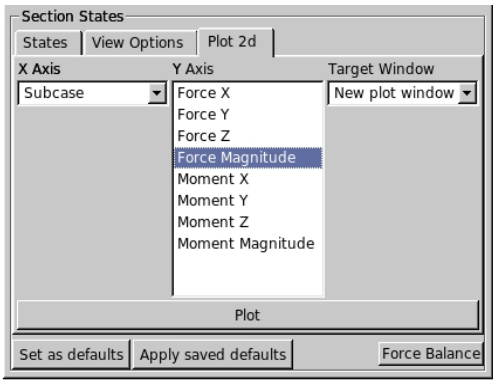
If necessary, grids can additionally be assigned to Sections through the *Add-Remove Nodes* tab:




SPC, MPC, Force, Interface and user-specified nodes can be assigned to a Section in one step.

Section Forces and Moments are automatically calculated by META by pressing the *Read Results* button. It is then possible to graphically visualise Force and Moment vectors through the *View Options* tab or plot the calculated Force and Moment results from the *Plot 2d* tab:



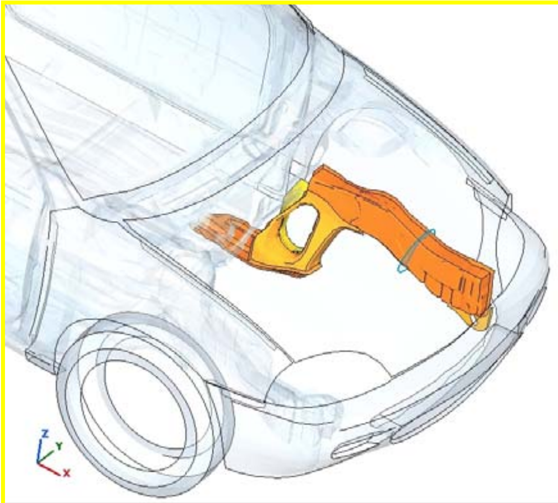

Using the above functionality the user is able to create a Section in any model region and accurately calculate Forces and Moments with accuracy comparable to those calculated by the solver within seconds. Thus, it is not necessary to define new Sections in the Pre-Processor and re-solve the Finite Element analysis, thus saving valuable time. Following is a verification of the accuracy of calculations performed in this field by META, against results extracted by the supported solvers.

## 2. VERIFICATION OF CALCULATIONS' ACCURACY ON CRASH MODELS

Scope

An **LS-DYNA** frontal crash model with sections specified in the input deck will be used for comparing the results between the solver and META.

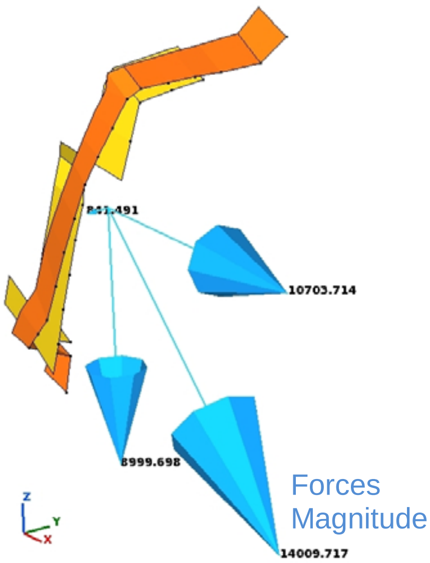
A Section in one of the model's front rails will be examined and the corresponding Force and Moment results will be calculated in META. These will then be plotted against the ones from the time history of the analysis for comparison.



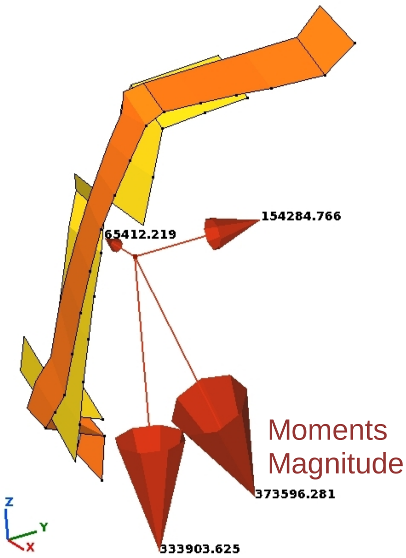
Verification

After loading the results, the Section Forces and Moments can be graphically displayed on the section by activating the corresponding options in the *View Options* tab:

Force components & magnitude display

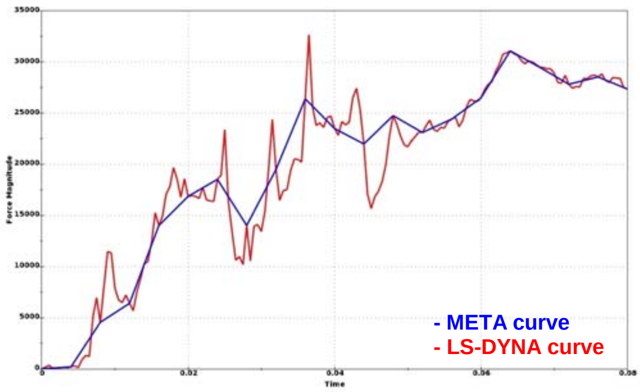



Moment components & magnitude display

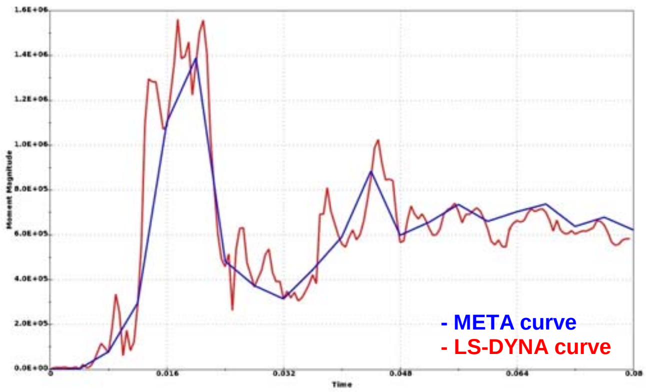



The vectors originate from the geometrical centre of the nodes assigned to the section. Subsequently, the Force and Moment results calculated by META are plotted against those of the LS-Dyna time history:

Forces plot

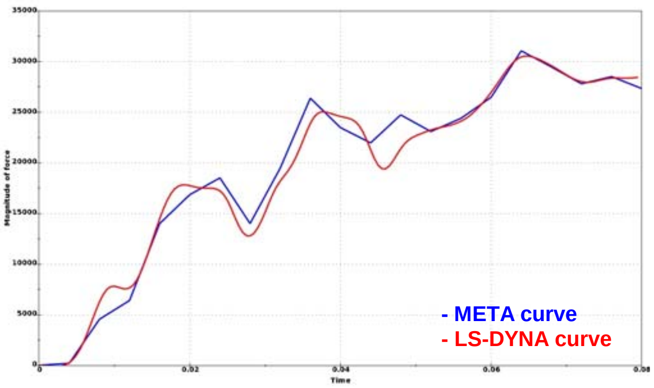



Moments plot

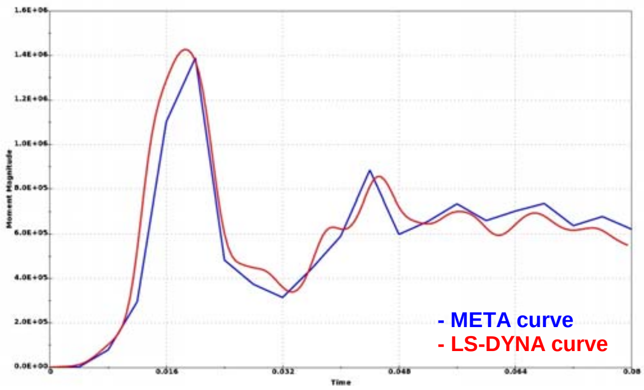


Some discrepancies between the two curves can be observed, however these are greatly reduced if the curves from the solver time history are filtered:

Forces plot - filtered




Moments plot - filtered



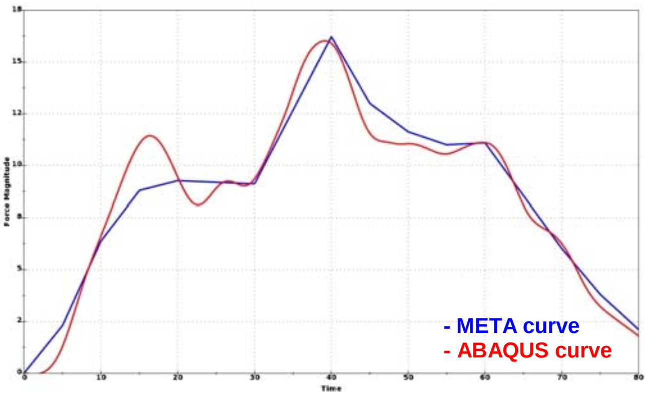

It is clear that the curves calculated by META closely follow those from the solver.
Some peaks missed out are considered to be acceptable as the time interval for the calculation of the curve from META is much larger than that of the time history curve.

Verification of calculations against other explicit solvers

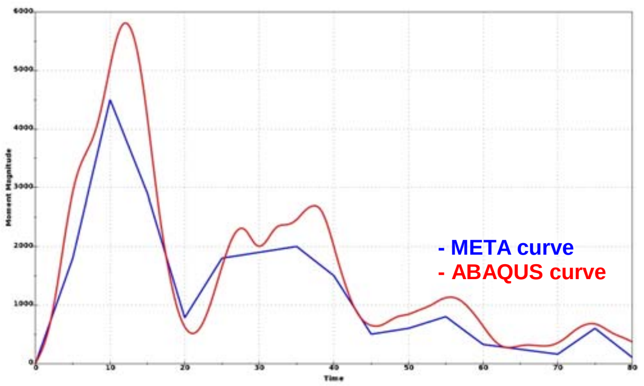
Following a similar approach, it can be shown that the results calculated by META are also accurate for the other supported explicit solvers. The graphs below demonstrate calculations from similar analyses, i.e. for sections defined in a model's front rail.

**ABAQUS Explicit**

Forces plot – filtered

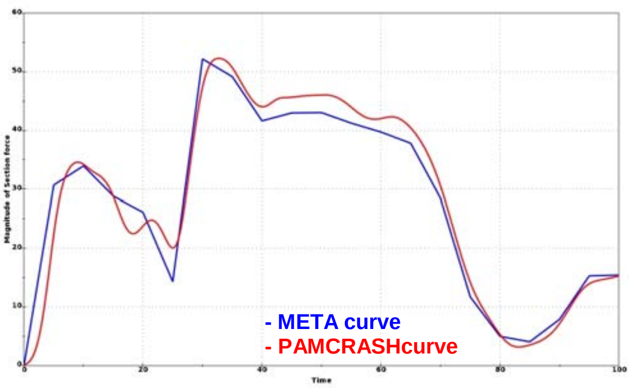



Moments plot - filtered

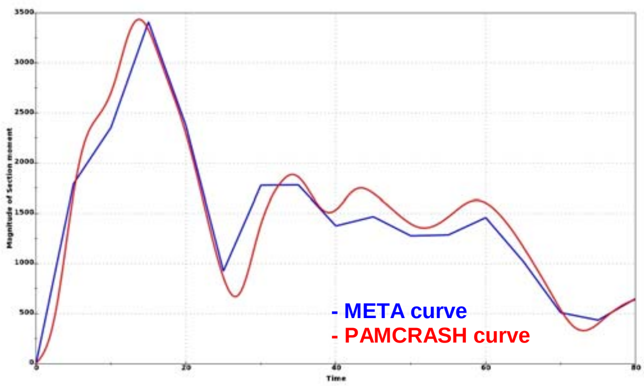



**PAMCRASH**

Forces plot – filtered




Moments plot - filtered



Having verified that the accuracy of the Forces and Moments calculated by META closely match those calculated by the solver, for existing sections, it is safe to conclude that this will be the case for Sections defined in META. This prevents the re-solving of crash models for sections not defined, making META a valuable Post-Solution calculator.

## 3. META FUNCTIONALITY FOR SUBSTRUCTURING OF DURABILITY MODELS

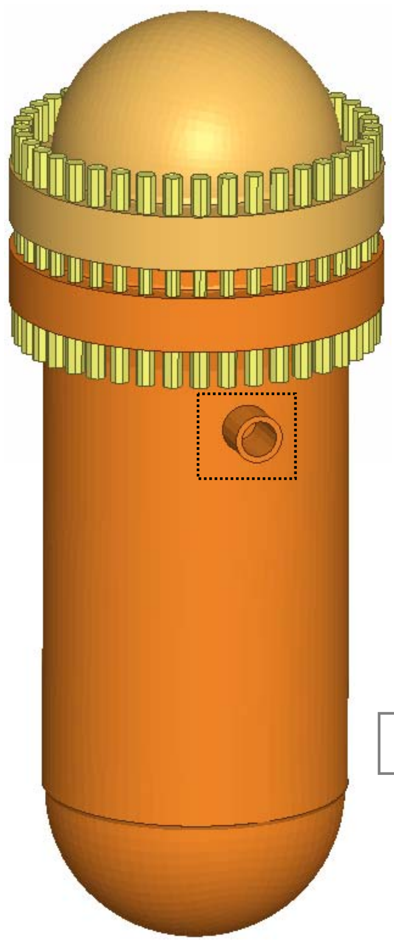
### General

Substructuring is a term used to describe the process of reducing the analysis of a large model to only an area of interest. Such a process, when set-up correctly, can help save valuable time and resources for analysis with little or no effect in results accuracy. The accuracy will depend on the correct definition of the Boundary Conditions (BCs) that will be used on the area of interest for replacing the rest of the model. The Section Forces tool of META offers functionality for the extraction of such Boundary Conditions and its effectiveness will be verified in the following pages through a simple substructuring process.

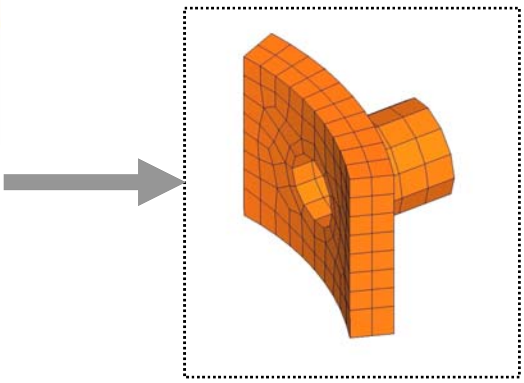
### The substracturing method

The workflow for the substructuring process presented in this paper is as follows:

The full model is solved with a coarse mesh → In META the area of interest is isolated → Grid point forces are calculated & saved for the outer nodes → In ANSA the model is reduced and META BCs are applied → Coarse & Fine meshed submodels are solved & compared

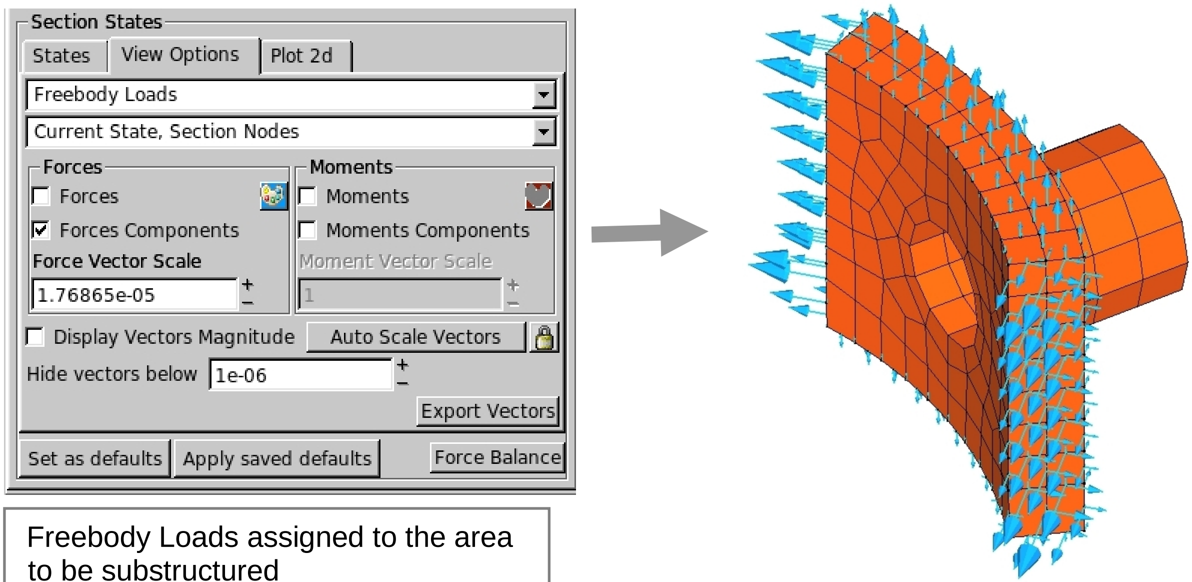


As an example, an ABAQUS-Standard finite element model of a reactor enclosure will be used.
The full model consists of 43780 elements.
The area of interest to be isolated is shown below and initially consists of 196 elements.
This will later be re-meshed with a finer mesh of 718 elements.



Full model and area to be substructed

Through the Sections tool of META, it is possible to calculate, from the full model's solution, the Freebody Loads on the external nodes of the isolated area of interest:

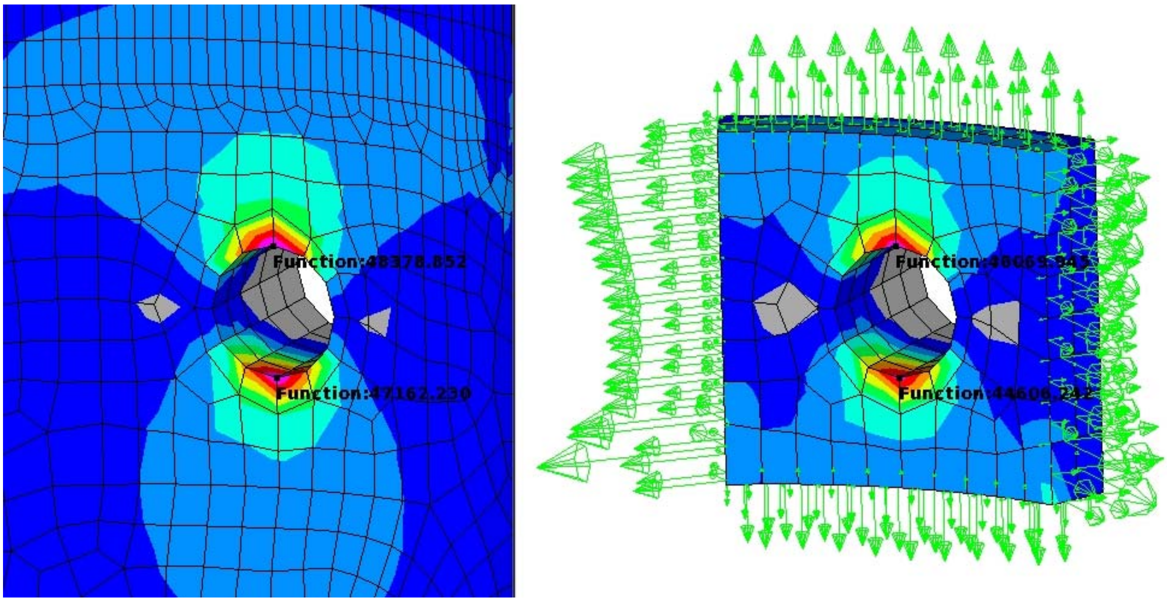



Freebody Loads assigned to the area to be substructured

These can subsequently be exported in NASTRAN bulk data or, in this case, in ABAQUS input file format through the *Export Vectors* button.

In ANSA only the area of interest is kept from the full model, the rest of the structure being replaced by the Freebody Loads exported from the Section Forces tool of META and assigned to the corresponding nodes.

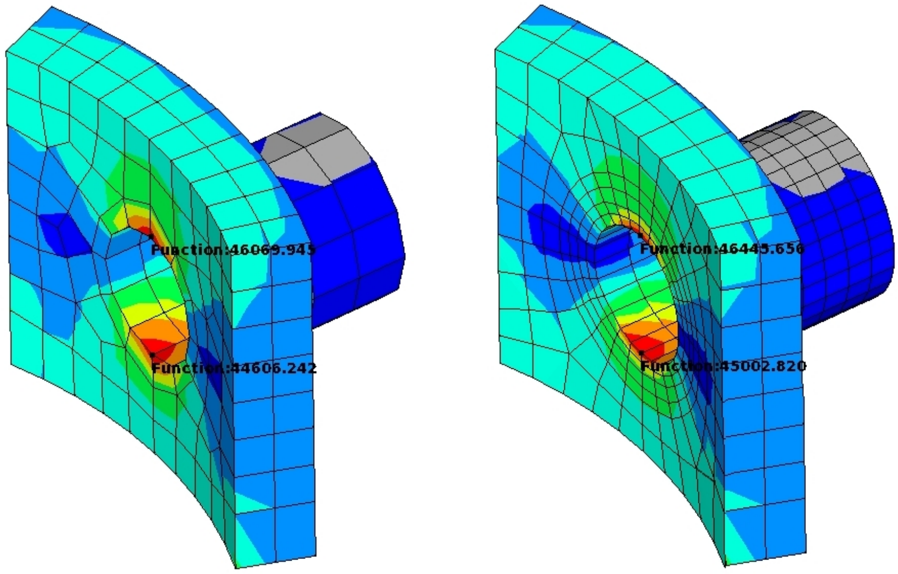
The mesh is deliberately left unchanged so that after the reduced model is solved in ABAQUS, its results are compared to those of the full model:




Solution comparison between full and substructured models

As it can be seen, the results closely match.

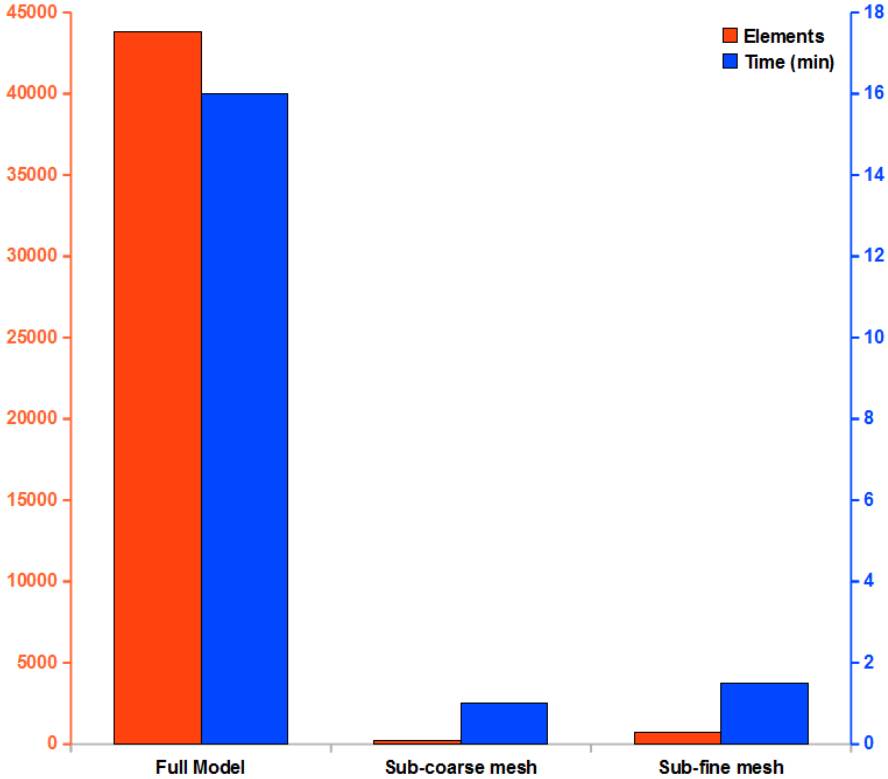
To conclude the substructuring process, since the accuracy of the exported force vectors as Boundary Conditions has been verified, the substructured area mesh is refined and the model is solved again:




Solution comparison between coarse and fine meshed substructured area

It is clear how the fine mesh gives a more accurate insight of the behaviour of the model compared to the coarse mesh model.

The bar charts below present the time and element number evolution of the aforementioned solutions: Full Model, substructured area with the same - coarse - mesh as the full model and substructured area with finer mesh:



## 4. CONCLUSIONS

The Section Forces tool of META is a valuable tool for calculating Forces and Moments on sections. Its calculation accuracy has been verified through the comparison of results from the Section Forces tool and from supported explicit solvers for pre-defined model sections. The accuracy was judged to be acceptable and thus meaningful in making calculations on new sections defined by the analyst within META, without having to repeat the task of defining them in the Pre-Processor and running again the analysis.

Another useful feature of the Section Forces tool is the option to export grid force vectors. These can be used as Boundary Conditions to accurately replace model areas of little or no interest to a durability analysis and thus provide a fast means of reducing the model size while in the same time gaining in calculations accuracy.

## REFERENCES

(1) μETA PostProcessor version 6.4.0. User's Guide, BETA CAE Systems S.A., July 2009
(2) LS-Dyna v971 Keyword User's Manual, June 2009
(3) ABAQUS 6.8 Analysis User Manual
(4) Three-Dimensional Submodeling of Stress Concentrations, J. R. Beisheim, G. B. Sinclair.