

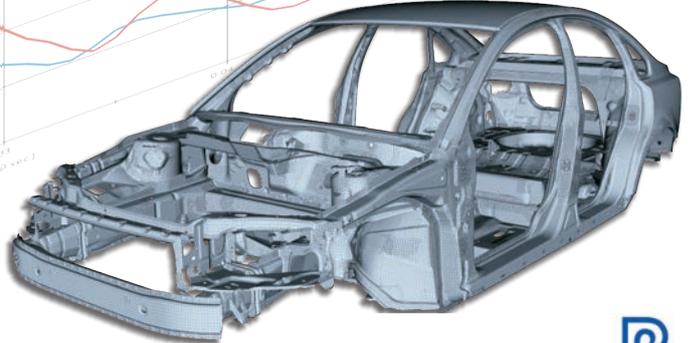
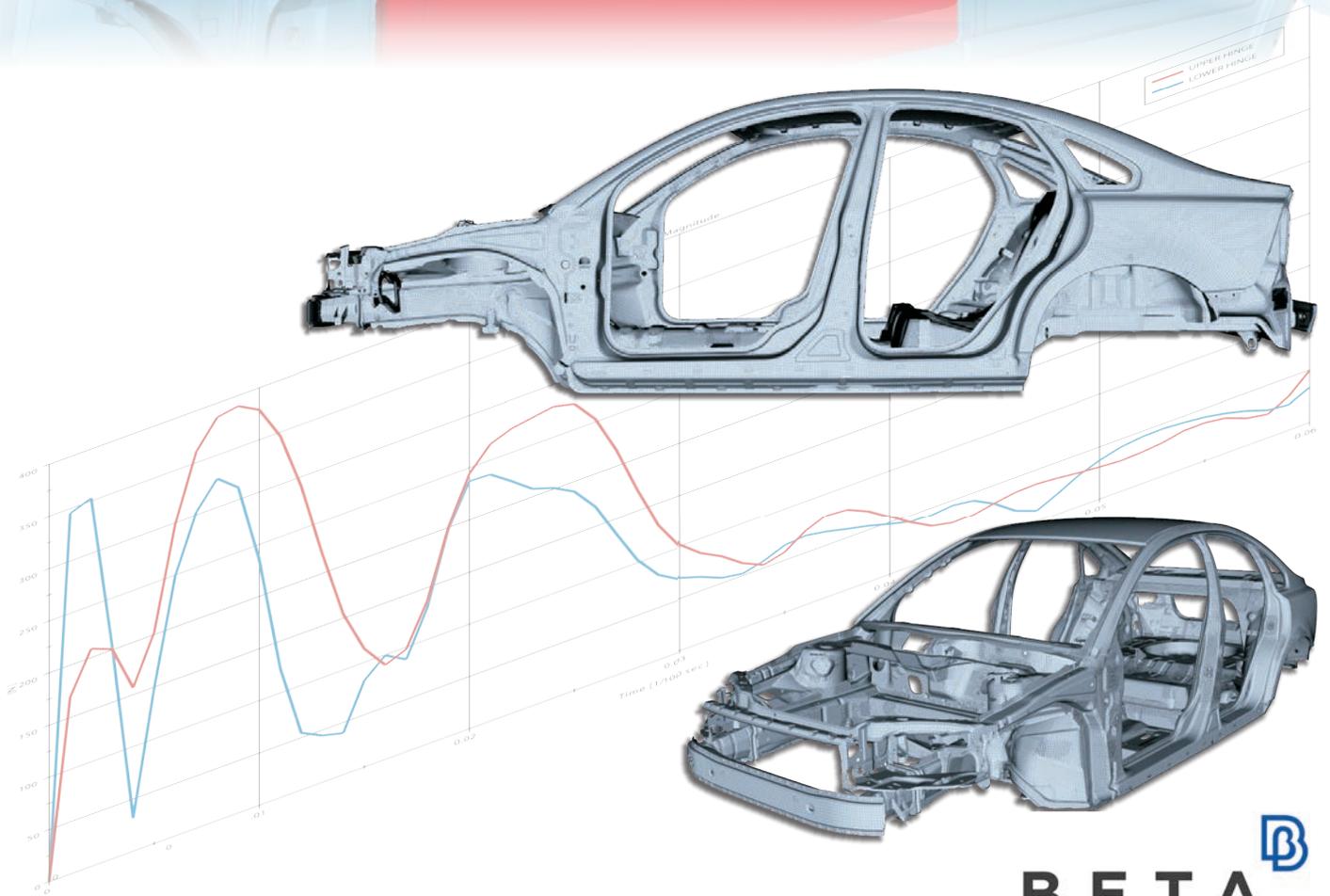
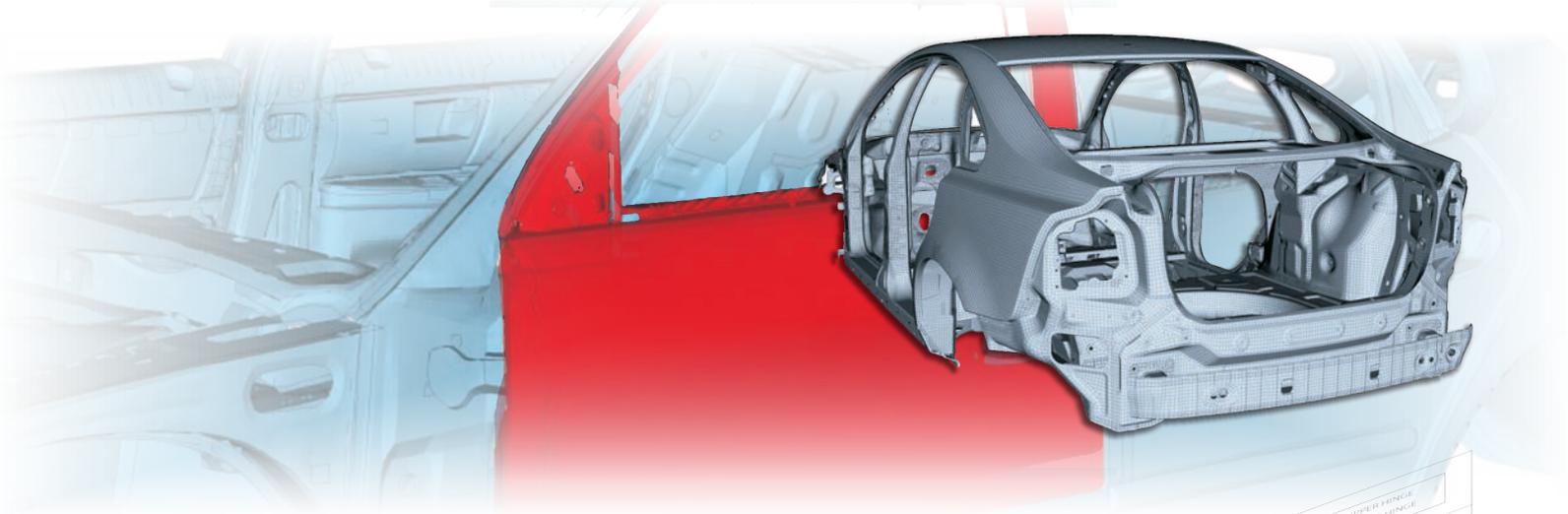
ANSA
PRE PROCESSOR



META
POST PROCESSOR



innovative solutions for
durability and fatigue
pre- & post-processing

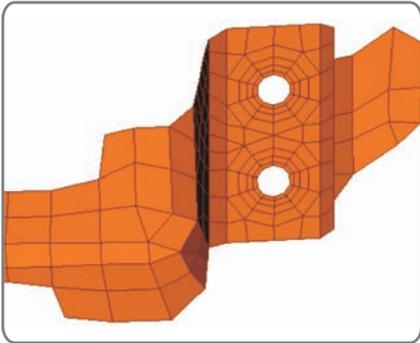


BETA
SIMULATION SOLUTIONS

www.beta-cae.com



ANSA is the only pre-processor capable of increasing the efficiency of the detailed FE-modeling for Durability, as it provides a variety of features suitable for demanding engineering tasks that meet the requirements of today's industry. It efficiently supports popular codes, such as Abaqus/Standard, ANSYS Mechanical, NASTRAN and PERMAS. The complete pre-processing environment hosts a broad range of functionalities. Especially when integrated with the Task Manager and the Data Manager, the pre-processing work acquires unique efficiency and repeatability during the build-up, update, and export of numerous high-quality models.

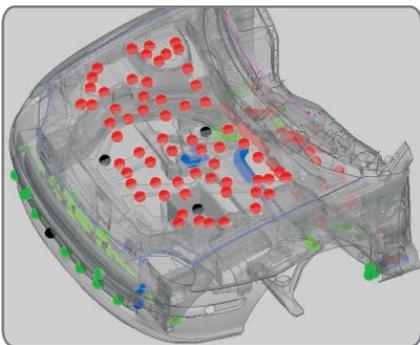


Meshing features

ANSA offers a wide range of meshing options and features adapted to durability specifications.

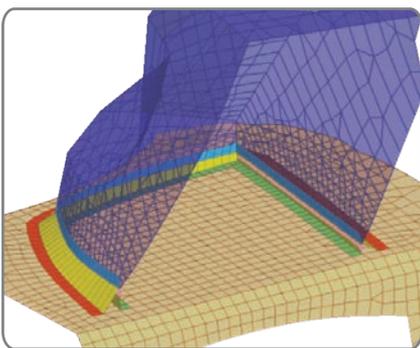
Proper local refinements in crucial areas - a common task of analysts - are achieved in a way such as to meet several durability requirements:

- Smooth transition between coarse and fine mesh areas
- Up to 15 zones around holes for more accurate discretization
- Proper mesh density with respect to durability analysis specifications
- Enhanced HEXA meshing with variable element length



Model assembly

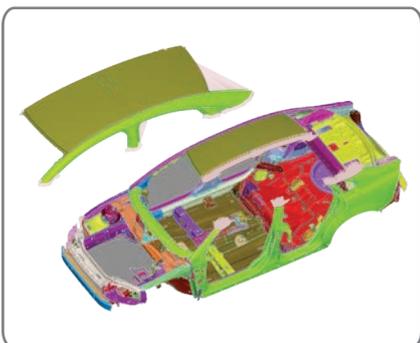
ANSA provides the capability of a full overview of the model, its components, and their changes. These may be visualized via multiple navigation methods, such as per parts, per include file, or per keyword. Moreover, the model part representations can be automatically controlled, even in case of multi-instantiated parts. Include Files, accompanied with a tree-like navigator, offer a successful management of the model composition. On the other hand, the Renumber Tool provides an overall control over the keyword ID's management, by enabling any potential numbering rules scenario. The decision making, as well as the application of model updates, is achieved with the help of the comparison tool, on parts or subassemblies level. This keeps the integrity of the model intact, even after the re-application of connections, boundary conditions, and mass re-distribution.



Weldings modeling

ANSA offers a wide range of multiple semi- and full-automatic tools for the creation of welding models. These models are based either on definitions, created within the software, or on information imported through PDM systems. The Connection Manager enables complete multiple connection type definitions in a single step, achieving:

- Broad range of weld definition types for spot welds, adhesives, bolts, solid bolts, Rob scans or seam lines
- Concrete position of connection representation
- Suitable element definition with controllable quality
- Detection & enhancement of unsuitable connections information



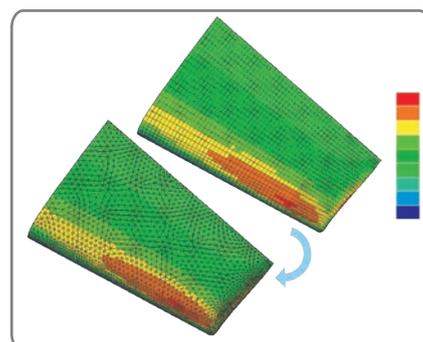
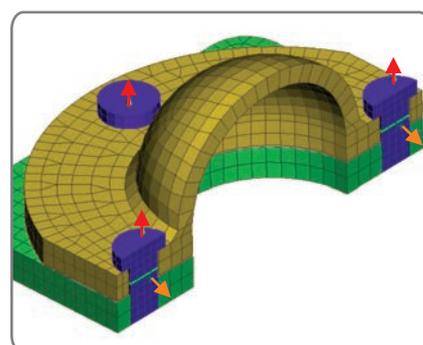
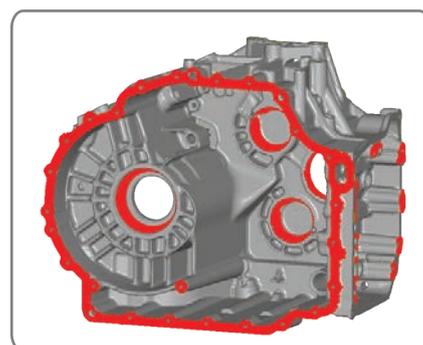
Model cut – substructure - submodel

Durability analysis models typically consist of an assembled BiW and a sub-system under loading. Quite often though, in order to reduce solution time, a problem is studied on a smaller scale. To address this, ANSA offers the "Model Cut" functionality. With the aid of this tool, the engineer is able to define an area of interest on the full body, isolate and properly mount this area, while preserving loadcase attributes of the original loadcase setup. The derived model is a clean, trimmed out part of the original, relieved from unwanted model entities. At the same time, it remains suitable for the generation and submission of a loadcase analysis.

As a special case of "Model Cut", ANSA can generate models for Substructure and submodel analysis. Utilizing the Substructure Generator tool, the user can define and manipulate such a procedure. The tool allows the manipulation of the generated data, the re-usability of the generated substructures and the easy creation of loadcase variations. ANSA offers all necessary keywords to built-up submodel analysis in a user friendly environment.

Contacts management

The contact detecting feature guides the definition of contact interfaces of areas of interest, with a preview and simple interaction. The massive modifications capability allows the fast modification of the properties of already existing contact models.

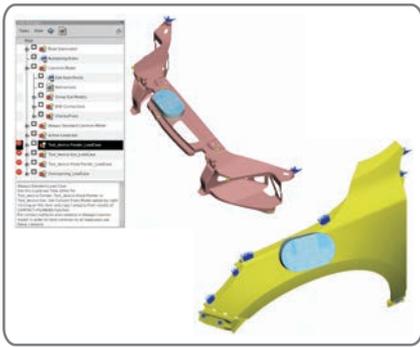


Features

- Interoperable decks
- Process automation
- Model assembly
- Include-files handling
- Mastering of entities' IDs
- Weldings modeling
- FEMFAT, FRMSIT, FE-SAFE, FDYNAM welds modeling
- Laminates Tool
- Positioning & Kinematics Tool
- Contacts detection
- Mesh refinement
- HEXA block meshing
- Arbitrary cross section calculator
- NASTRAN SOL 400 set up
- Substructuring/Superelement
_Submodeling
- Results mapping
- Quality validation & fixes
- Pre-tention assistant
- Analysis manager
- NEF (NASTRAN embedded Fatigue)
- Results based refinement
- Numerous solvers supported: Abaqus, ANSYS, PERMAS, LS-DYNA.

Benefits

- Cost and time-to-market reduction
- Novel features lead faster to results, while ensuring efficiency and quality
- Automated tools for the definition of the modeling steps of durability analysis

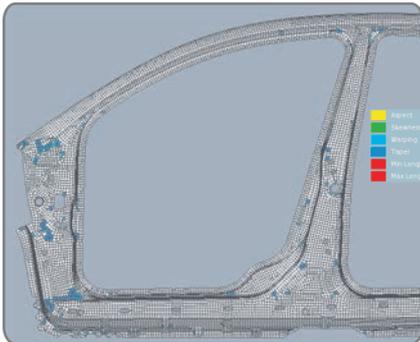


Pretension Loads

The Pre-tension definition is comprised of a stepwise tool where the vector and section are prescribed with minimum user interaction. Especially, the Pre-tensions of bolts passing through the same plane are recognized and generated in a rapid way.

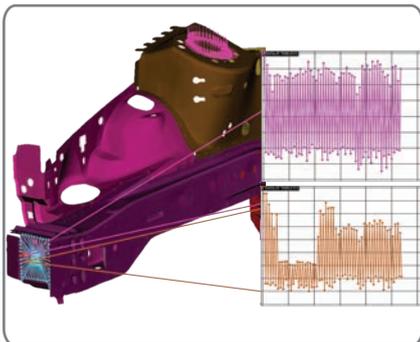
Results mapping

Data regarding nodal thickness, pressure, initial stress etc. can be mapped from an existing file to a different mesh. The source file can contain a variety of result types, such as information regarding thinning that derives from stamping, pressure and temperature results of a CFD analysis that will be the loads of a durability model or initial stress results to be considered for local mesh refinement. Any type of result can be mapped through any user defined ASCII format types.



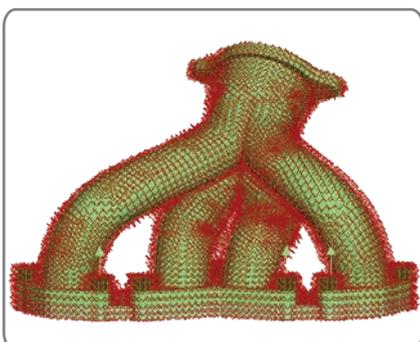
Task manager - Load cases definition

The Task manager module organizes the sequence of steps and actions that capture the generation of a model build-up. It automates the process and safeguards model quality by capturing actions that need to be repeated by the analyst. Also, it allows for the easy and repeatable generation of loadcases as they are mandated by regulations. Actions such as mesh refinement, assembly, model cut, contacts' definition, indentors positioning and variable loading can be parametrized and seamlessly repeated. GEBS are exploited to facilitate the parameterization of the modeling steps.



Quality checks & improvements

Based on solvers' quality criteria and thresholds, ANSA provides a wide range of checks and improvement algorithms for model's quality and integrity. Dependency issues, improper Contact & Tie definitions, poor element quality or loose parts are recognized and reported. Most of these issues are fixed in an automatic manner. Detailed mass information is calculated and reported, always in full compliance with the solver. ANSA scripting language can be employed for the further definition of custom quality checks and fixes. Checks can be template driven in order to repeat them for any model in a fast manner.



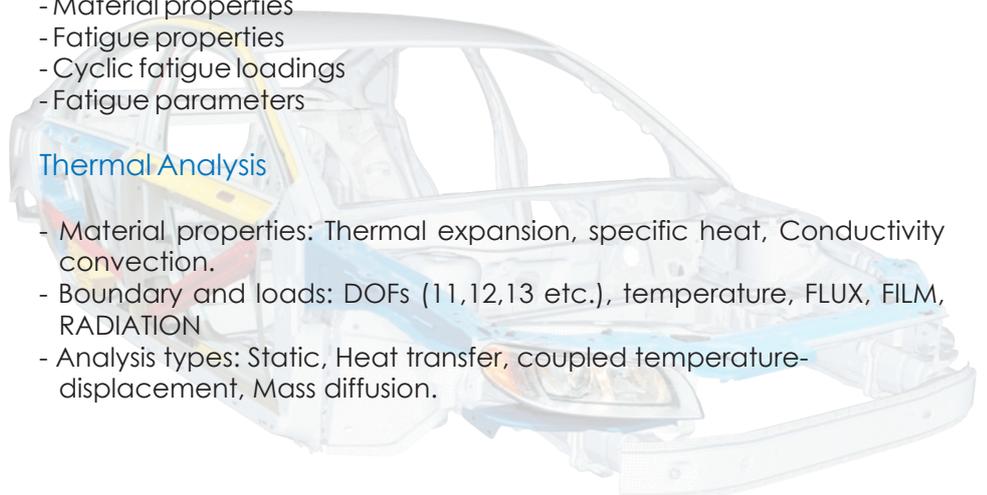
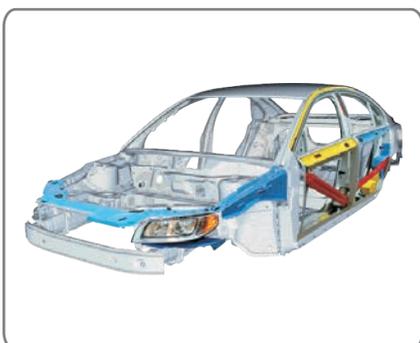
NASTRAN Embedded Fatigue

ANSA offers a single environment for NEF set up which accomplish the following steps of the process:

- Initial model set-up
- Material properties
- Fatigue properties
- Cyclic fatigue loadings
- Fatigue parameters

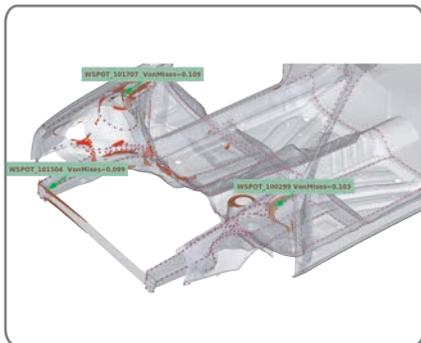
Thermal Analysis

- Material properties: Thermal expansion, specific heat, Conductivity convection.
- Boundary and loads: DOFs (11,12,13 etc.), temperature, FLUX, FILM, RADIATION
- Analysis types: Static, Heat transfer, coupled temperature-displacement, Mass diffusion.



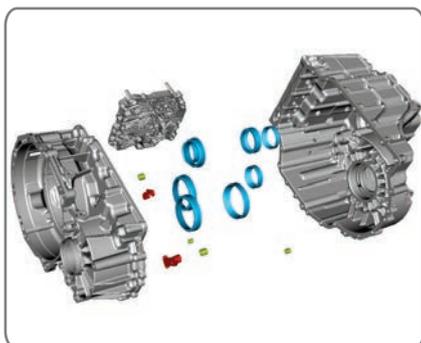
META is a highly sophisticated yet user friendly post-processor. Through its continuous development it has succeeded in meeting the ever increasing demand for efficient and thorough post-processing. META supports results from all popular static and fatigue analysis solvers, as well as real-life test data and can even be integrated in optimization processes.

Taking advantage of the unique window configuration options and the easy handling of multiple models data analysis becomes more efficient than ever. Through its top quality 3D graphics and high performance, post-processing becomes better and faster than ever.



Model & connections management

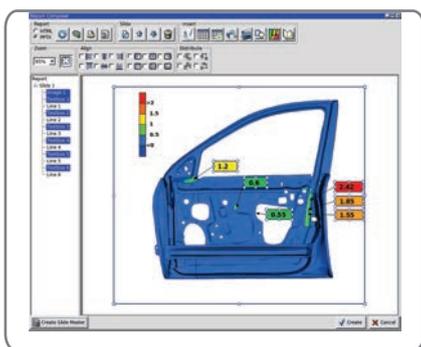
Smart and intuitive main functionality with numerous options succeeds in managing and manipulating the model requiring minimum time and effort. Results from contact analyses are conveniently displayed and efficiently handled through a list. A complete sets tool advances model handling capabilities to a high level through a tree structure hierarchy. On top of that, spotwelds can also be represented and managed or updated easily per type & per property level, hence their post-processing becomes an easy task.



Hot spots identification, communication of results & reporting

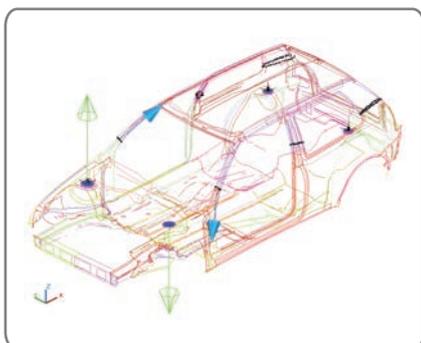
META has versatile filter capabilities incorporated in many of its tools. Filters can be used alone or combined based on results, names, position, regions and many other criteria to identify, isolate or pin-point areas of interest (hot spots). An overview of these areas' results is achieved through statistics tables with spreadsheet functionality. These tables also provide an efficient means for fast comparison of results from different iterations. Run-time annotations that trace key areas of the model and easy-to-create reports in html, PostScript or MS Office PowerPoint .pptx format further boost communication of results.

Interactive run-time creation of reports is easier than ever by dragging/dropping images and copy-to-clipboard functionality for transferring data. Moreover, reports in pptx format can also be input in META, modified and previewed in slide show mode.



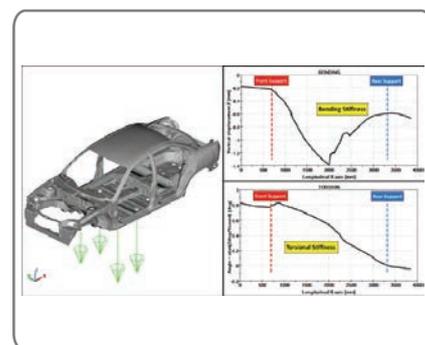
Results calculation

Addressing specific needs of durability and fatigue analysis, META features a calculator for the linear combination of results deriving from other loadcases. New datasets can be created by applying any mathematical operation on existing data of either the same or different models. Results can be mapped from one model to another, so as to facilitate the comparison between non-compatible meshes. Whenever required, data can be easily scaled or transformed with respect to a local coordinate system. Forces and moments on any user defined section can be calculated and can be output in solvers format along, with the enforced displacements, to be used to sub-modeling. Strain Gauges can be created and have results calculated from the dedicated tool. Stress linearization results can be calculated and plotted on the areas of interest and as curves, while a report can be created for an overview of the results.



2Dplots

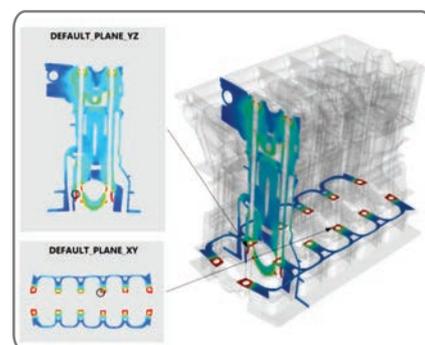
Direct correlation of simulation and experimental data both in 3D and 2D fields is significantly simplified without the need to start a separate software. A powerful graph tool enables the direct plotting of data deriving from either the 3D model or from data imported from a wide range of time history files, including .iso and .unv format. Furthermore, any arbitrary line-path can be easily defined through a sophisticated feature selection assistant and the results along this line-path can be displayed in 2D plots. Through the wide array of tools, curves can be traced in relation to the 3D model (subcase after subcase), compared, mathematically manipulated and annotated.



Output options

Images and videos of 3D and 2D results are saved in all popular formats (png, tif, jpeg, ps, avi, mpeg, gif and many more). Among other outputs, deformed geometry, cross sections and iso-contours can be exported in solver's format.

In addition, model data as well as other post-processing information is stored in META native binary files which can also be compressed and viewed with a license-free viewer.

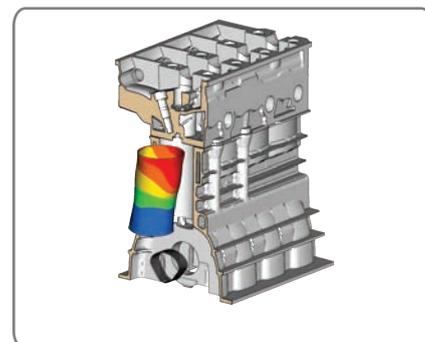


Automation

Post-processing for durability and fatigue analysis is greatly assisted by parameterized sessions and scripts which provide the grounds for the automation of long and complicated procedures, such as stiffness analysis, or bore distortion analysis, thus boosting productivity and assuring the quality of final report. Furthermore, the coupling of META with external optimisers is achieved through the use of a toolbar provided with the software.

Models comparison is simplified and accelerated through repeating actions previously recorded for the model on another model.

The automation capabilities take advantage of the Python scripting language, well adapted for engineering applications.

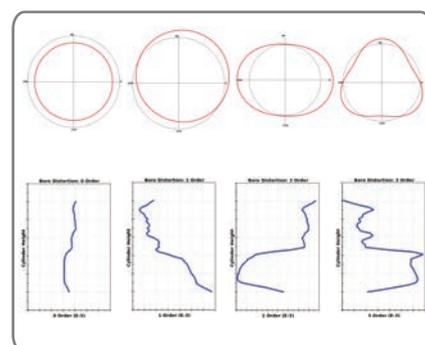


Features

- Fatigue results
- Connections handling
- Filtering & queries
- Hot spots identification
- Statistics & spreadsheet
- Results calculation
- Linear combination
- Section forces calculation
- Report composing
- Bore Distortion Analysis toolbar
- Stiffness Calculation toolbar
- Toolbar for composites post-processing

Benefits

- Reduced post-processing time
- Effortless model handling
- Rich pre-information availability
- Flexibility through filters use
- Concurrent 3D and 2D processing
- Ready-to-show report creation
- Easy set-up of reliable automated processes
- Novel features lead faster to results, while ensuring efficiency and quality



BETA CAE Systems International AG

D4 Business Village Luzern, Platz 4, 6039 Root D4, Switzerland
T +41 41 545 3650, F +41 41 545 3651
ansa@beta-cae.com, www.beta-cae.com

© BETA CAE Systems International AG 2017

Features subject to change without notice.
All trademarks are property of their respective owners.
Printed in Greece.