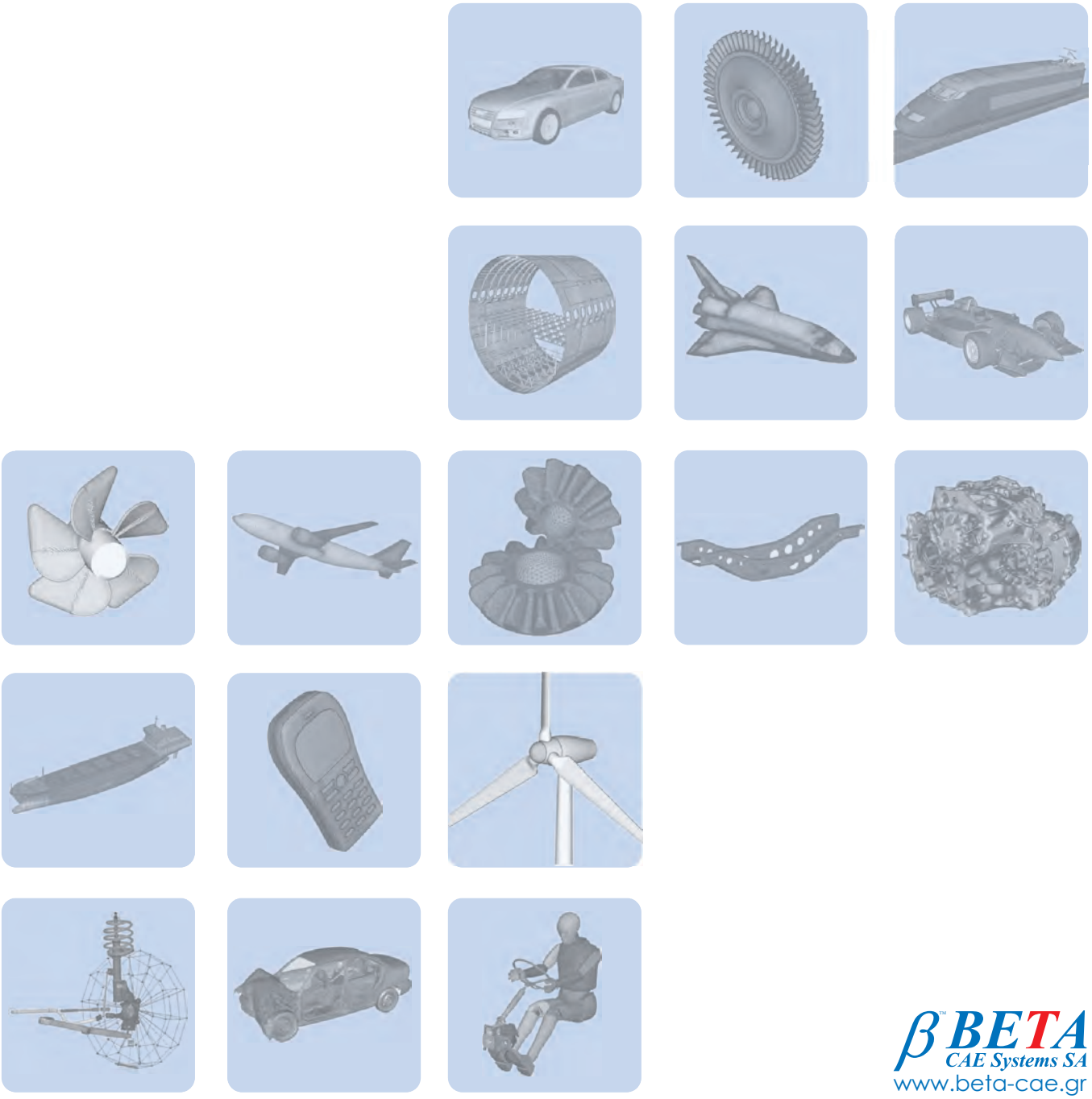
ANSA
μETA
PostProcessor

pioneering
software systems

for multidisciplinary CAE pre- & post-processing



BETA
CAE Systems SA
www.beta-cae.gr

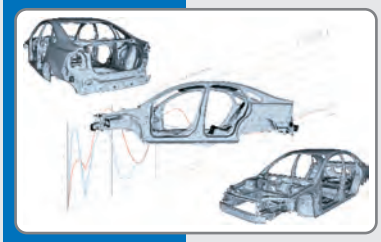
multidisciplinary CAE pre- & post-processing

ANSA is an advanced multidisciplinary CAE pre-processing tool that provides all the necessary functionality for full-model build up, from CAD data to ready-to-run solver input file, in a single integrated environment. ANSA is the users' preference due to its wide range of features and tools that meet their needs.
μETA is a thriving multipurpose post-processor that meets diverging needs from various CAE disciplines. Combining its highly sophisticated capabilities, the user friendly interface, and its automation capabilities μETA facilitates the effective and thorough post-processing. Additionally, its impressive performance, its innovative features, and capabilities of interaction between animations, plots, videos, reports and other objects, makes μETA a complete solution for fast and advanced post- processing.

quality and performance in automatic mesh generation

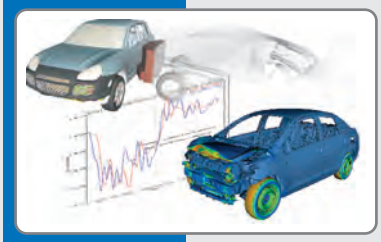
ANSA, combining the automatic geometry healing with the automated middle skin extraction and the Batch Meshing, offers the most efficient creation of meshed models to even the less experienced users. The resulting shell and volume elements models easily meet the meshing requirements and quality criteria in less than 75% of the time needed by other software. The Batch Meshing Manager tool is completely integrated into ANSA and allows for the detailed definition of meshing scenarios.

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



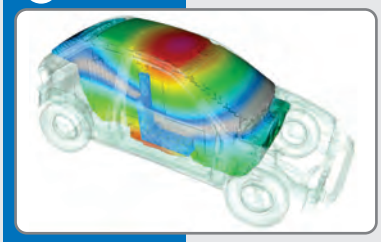
ANSA is a pre-processor capable of increasing the efficiency of the detailed FE-modeling for Durability, as it provides a variety of features suitable for demanding engineering tasks that meet the requirements of today's industry.
Through its continuous development, μETA has succeeded in meeting the ever increasing demand for efficient and thorough post-processing. Due to the unique window configuration, the easy handling of multiple models data, its top 3D graphics, and its performance, post-processing becomes more efficient than ever.

the standard in Crash & Safety pre- & post- processing



ANSA is the Industry standard solution for crash and safety pre-processing. The software's powerful functionality allows users to perform the required tasks in less time and at lower cost. The innovative ANSA concept of interoperable decks allows for the easy conversion of a model from one solver input deck to another providing supreme flexibility.
μETA stands up to the modern crash-analysis challenges, such as the increasing model size and the numerous model comparisons. It features a powerful graph tool, a broad range of build in functions for 3D and 2D post processing, multiple statistics windows, and efficient memory usage that facilitates the comparison of large models.

advanced solutions for NVH pre- & post- processing



ANSA is a standard and robust solution for NVH simulation pre-processing that addresses the industry's needs for increased process efficiency and simulation results reliability. The interoperable pre-processing decks, integrated with the advanced main features, such as those of meshing, batch meshing, morphing, and coupling with optimizers, comprise a complete environment that accommodates a wide range of functionalities.
μETA brings a new dimension to the NVH post-processing by successfully addressing the bottlenecks and constraints involved. Its optimized performance for graphics memory usage and processing time along with a broad range of features and calculation options makes it an effective environment for handling all NVH post-processing requirements.

for demanding CFD pre- & post-processing

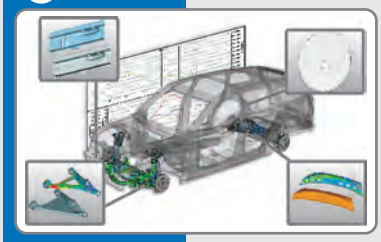
ANSA with its powerful functionality provides high efficiency solutions for CFD applications. Its capabilities meet the industry's demanding needs for external and internal flow simulations, increase productivity, and contribute to the high quality of CFD results. It is the choice of the leaders in CFD simulations in various sectors such as automotive, motorsports and aerospace among others. μETA, the leading post-processor in structural analysis, extends its support to CFD codes. μETA's indisputable high performance capabilities enable engineers to easily handle and explore extremely large and complex models. Through the numerous validated analysis tools and automation capabilities, engineers avoid time-consuming data mining and focus their engineering judgement on important facts.

optimum pre- & post- processing for powertrain

With ANSA's complete solutions for meshing, assembly, contacts definition, and boundary conditions set-up, ANSA becomes the most effective solution for powertrain pre-processing. ANSA satisfies the most demanding powertrain CAE processes through its easy to set up meshing algorithm and the powerful assembly center for the generation and handling of bolt models.
μETA is a complete solution for powertrain post- processing. Its model organization capabilities, the variety of tools for the assessment of the results, the functionalities for the identification of areas of interest, the options for the communication of the results, and its automation capabilities makes μETA the stand-up choice in the industry.

CAE tools for products made of composite materials

Composite materials demonstrate significant advantages, such as weight saving potential, increased strength and greater design flexibility. However, these are combined with significant design restrictions and considerations. In order to fully exploit the composite materials potential, cost and time effective design, analysis, and manufacturing processes are required. New tools are offered for the modeling, simulation, and analysis of the behavior of products made of fibber reinforced laminated composites. The Laminates Tool with additional special features in ANSA and the Composite Post Toolbar in μETA combined with the rest of the high performance functionality, offer a unique solution for products that include parts made of composite materials.

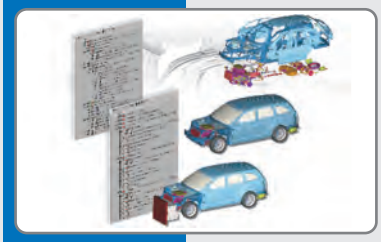


prerequisite for effective optimization

ANSA pre-processor and μETA post-processor, in combination with all popular optimization codes, provide a complete tool for optimization applications. From concept design to final testing, ANSA & μETA package brings enormous performance and versatility to optimization problems set-up. The ability to control the model shape using the ANSA Morphing Tool, the ANSA model values, and even complicated tasks such as the Batch Meshing and Model Checking, makes the tool unique.

facilitating data management in ANSA

Data management is unquestionably one of the most critical factors that contribute to the efficiency and productivity of CAE teams. The huge amount of data involved in pre-processing as well as the need for the synchronization of the CAE model with the design evolution, intensify the need for flexible and inexpensive solutions that can bridge the gap between the PDM systems and the CAE-world and become a reference point throughout the CAE cycle. Addressing these needs, BETA CAE Systems SA provides integrated solutions for the effective and cost-efficient management of data, starting from the PDM systems and all the way to the output of the keyword file.



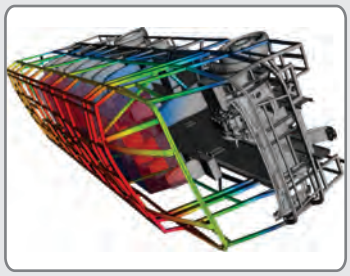
pioneer solutions for process automation

Process automation capabilities of ANSA can accurately capture any pre-processing task making any quality criteria and requirements inherent to the process. As a result, the productivity of engineering teams is improved and the quality of models is assured, leading to a drastic reduction in the CAE turnaround time.
μETA provides the analyst with a unique range of powerful tools to set up automated 3D and 2D post- processing tasks. Special action based language (session), Python programming and user defined toolbars can boost productivity offering post- time reduction, limitless automation capabilities and reliable repeatability.

specialized tools

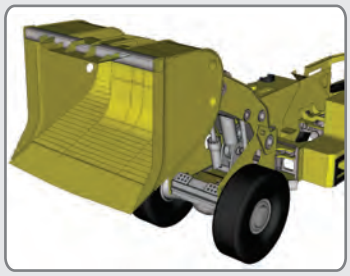
The Cross Section analysis tool offers features, such as the Cross section definition, the check and modification, the Computation of Cross-section geometrical results, the Cross section Stress analysis for variable loading points, and the Cross section edit card that can be used by script functions.
The Fuel tank analysis tool is a rapid simulation of the filling or emptying process of a closed tank with features such as the Volume vs Fuel level graph, the automatic volume-traps detection, the filling and suction points definition and analysis, and the real filling process monitoring.

- automotive



- trucks & buses

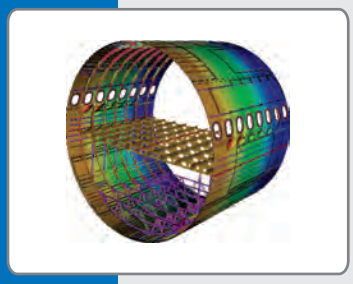
- motorsports



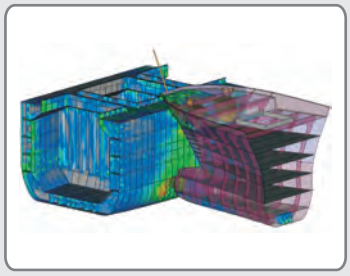
- heavy machinery

- rail vehicles

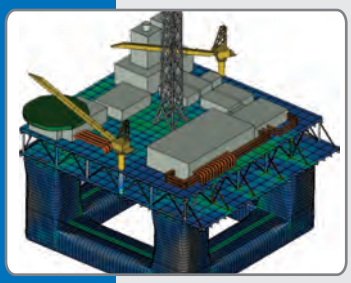
- defence



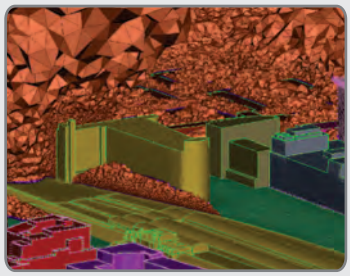
- aerospace



- maritime design

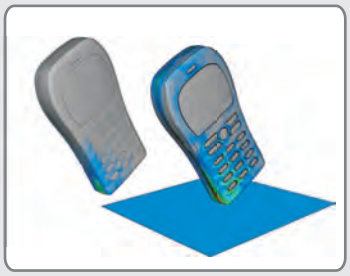


- offshore



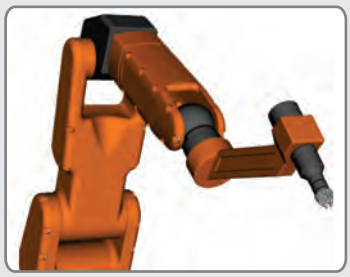
- architectures & construction

- energy

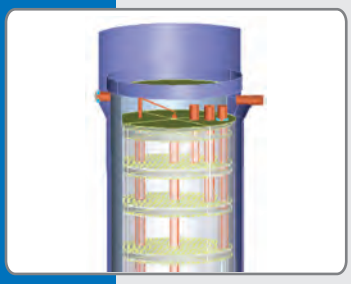


- electronics

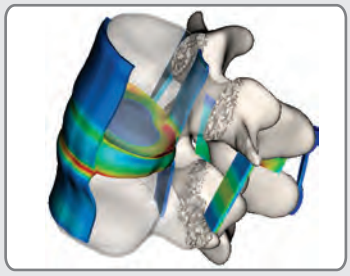
- power tools



- robotics & industrial equipment



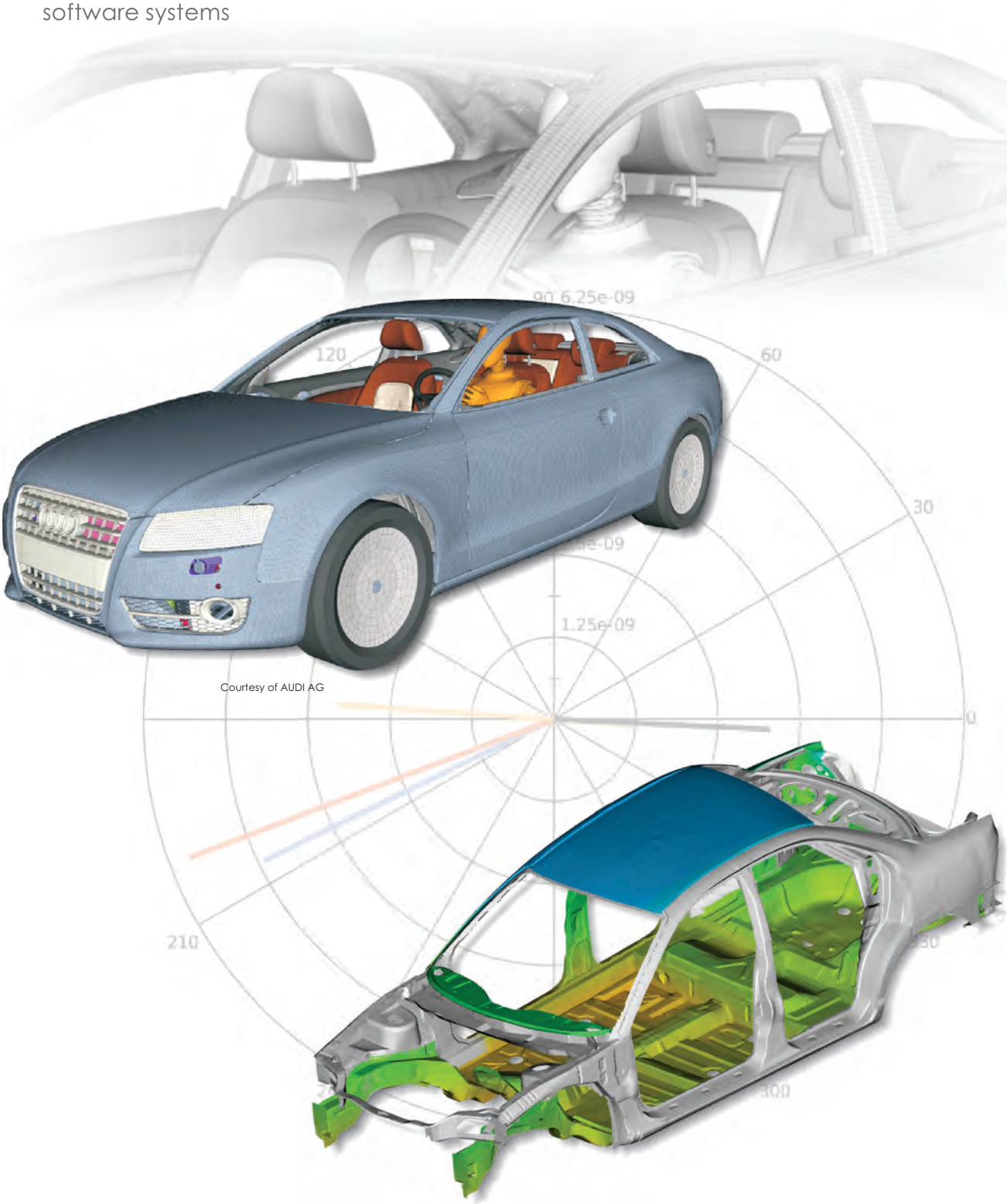
- processes engineering



- biomechanics

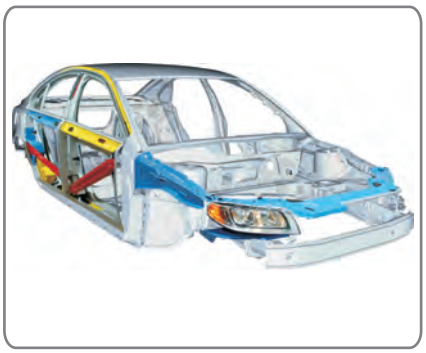
ANSA
µETA
PostProcessor
pioneering
software systems

for multidisciplinary CAE pre- & post-processing



Courtesy of AUDI AG

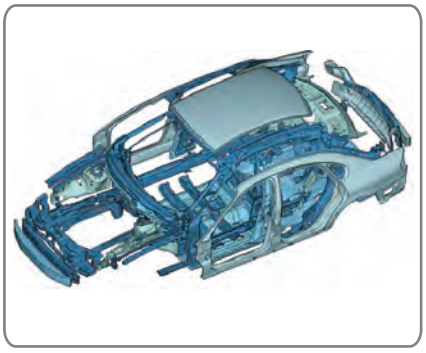
ANSA is an advanced multidisciplinary CAE pre-processing tool that provides all the necessary functionality for full-model build up, from CAD data to ready-to-run solver input file, in a single integrated environment. ANSA is the users' preference due to its wide range of features and tools that meet their needs. The list of productive and versatile features is long and the alternative tasks and processes that can be completed using these features are countless.



Courtesy of Volvo Cars Corporation

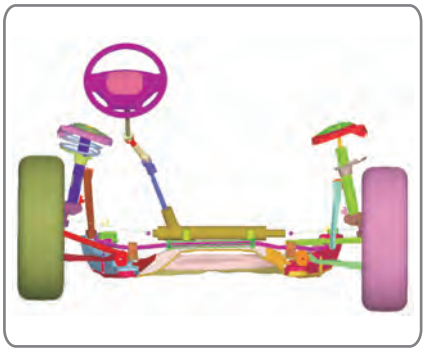
Environment

All software features are accommodated in an integrated environment, with highly customizable GUI. The software is available for all contemporary popular operating systems in 32bit and 64bit architecture with multi-core CPU usage. The accelerated graphics, the rapid confirmations and function access, the GUI customization options, the model browser and lists handling, the filtering and modification operations, and the integrated search engine comprise a user friendly environment that ensures outstanding performance and productivity.



CAD data input & clean up

CAD definitions and model structure data in CATIA V4, CATIA V5, NX, PTC Creo (Pro/ENGINEER), Inventor, SolidWorks and JT formats can be converted into ANSA files using the available translators. Moreover, custom Interfaces to PDM or SDM systems, powered by scripting, bring product & model structure data into the heart of the software.
CAD geometry can be also read in from neutral file formats (iges, step, vda-fs), manipulated and healed by the proprietary powerful built-in geometry engine. A wide range of geometry healing functions, including those for the generation of neutral fibers, deliver geometry descriptions ready to be meshed.



ANSA data management

ANSA Data Management (ANSA DM) is a centralized data management system, used to collect and store, in a structured and hierarchical form, all engineering data that are used during the development process of a vehicle simulation model. It assures the effective and efficient data handling throughout projects, by streamlining updated model data to engineering teams, allowing the easy sharing of common data and offering access to library items for the analysis dependent solution settings. The DM Browser, moreover, allows the browsing of the DM Root to identify the available CAD versions, study versions and representations for comparison and model update.

Model Comparison and update

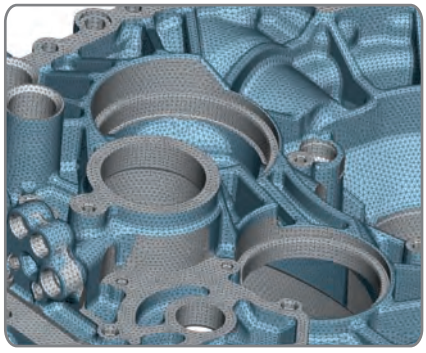
An integrated tool that compares two models in order to identify differences in geometry, attributes, solver-specific definitions, as well as connections. User friendly navigation and identification features are provided while a complete or partial replacement can be performed, updating the model according to user directions.

Seat model courtesy of BMW AG

Process Automation

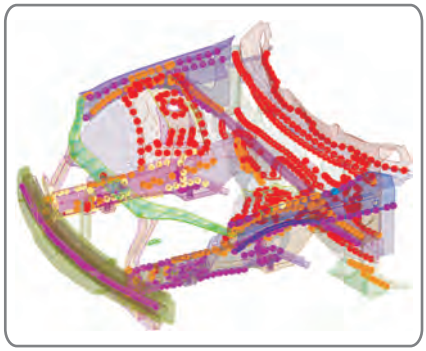
Task Manager and Scripting language provide a unique modeling solution for automated and effective applications.

Task Manager is an integrated workflow manager that includes all individual tasks of a simulation model development. The process template is built up by the CAE expert who sets the boundaries between distinct modeling actions and predetermines all modeling parameters that must be respected, leaving to the inexperienced user a minimum degree of interference and limited decision making.
The scripting language is an enhanced programming tool that boosts productivity providing the power to access data and perform custom operations in an automated way.



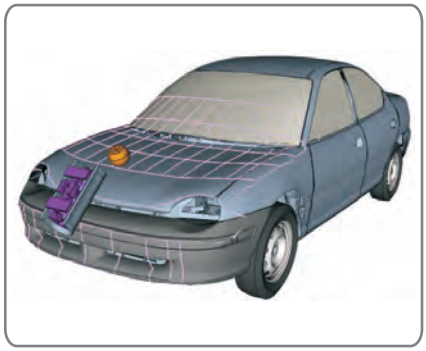
Meshing

The integrated Batch Meshing tool leads to controllable and effortless optimal results, for both shell and volume meshing. Following the versatile mesh area idealization, geometry can be meshed according to modeling requirements by cutting edge surface and volume meshing and wrapping algorithms.
A unique mesh generation environment is composed by:
- proprietary shell meshing algorithms
- high performance and quality volume meshing algorithms
- state-of-the-art boundary layers elements deployment
- Hexahedral dominant meshing
- Acoustic Cavity mesher and the straight forward Wrapping tool
- one-step mesh generation on automatically extracted middle surface
- numerous mesh handling functions.



Assembly

Powered with fully comprehensive parts and welding management tools, accommodates parts assembly, with alternative node-dependent or independent connections types, appropriate to various disciplines. Interfaces to numerous connections data file formats allow the completion of a single stage assembly. New concepts have been introduced, including model hierarchy input, multiple part instances handling, parts comparison-replacement and update as well as special joining type creation.

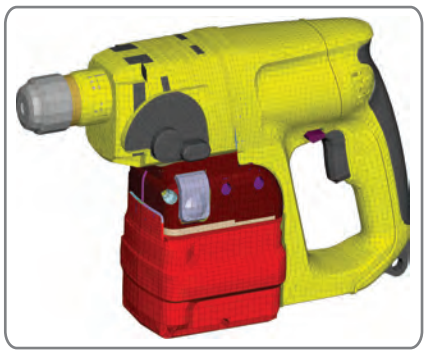


Include files configurator

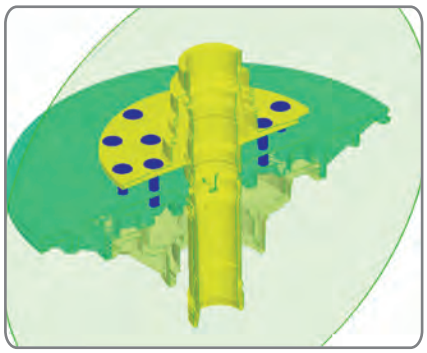
Different model variants or load cases can be compiled with available Include files through one of ANSA's special tools. This enables the creation of particular model configuration even without loading the respective data.

Pre-processing decks

Pre-processing completion is achieved through the uniquely interoperable pre-processing decks for NASTRAN, LS-DYNA, PAMCRASH, RADIOSS, Abaqus/Standard, Abaqus/Explicit, PERMAS and ANSYS Structural, allowing direct model modification between solvers, including material synchronization. Numerous unique utilities facilitate laborious tasks, such as the management of Includes Files, model sub-structuring, entity numbering control etc. ANSA is multidisciplinary by design, in order to simultaneously handle models for Crash, Durability, NVH analysis, and many other disciplines, supporting all entities required by the latest versions of solvers.
A compilation of CFD oriented features are accommodated into special CFD pre-processing decks that support the most popular codes, such as STAR-CD & CCM+, Fluent, OpenFOAM.

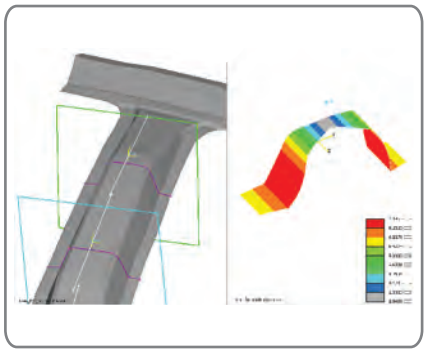


Courtesy of DeWALT Industrial Tool Co. and BAYER AG
Meshed by LASSO Ingenieurgesellschaft mbH



Analysis tools

Crash and safety modeling is assisted by user friendly features for impactors positioning, seatbelt fastening, positioning and articulation of crash test dummies and "headform" models for passenger and pedestrian safety simulation standard scenarios.
The fast and easy Kinematics tool solves sophisticated positioning problems, for seats, convertible roofs and other mechanisms.
The Laminate Tool is one more enhanced function that assists the modeling of complex parts made of composite materials.
Results Mapping tool enables the import of data, such as nodal thickness, pressure distribution, stresses and plastic strain, nodal temperature, material orientation, that can be mapped from an existing file to a different mesh. The tool ensures the accuracy in the mapping process and handles all major results.



Model integrity checks

Replicates the integrity and correctness checks performed by the solvers, reports potential modeling flaws and proceeds to model auto-correction actions.

Solution control

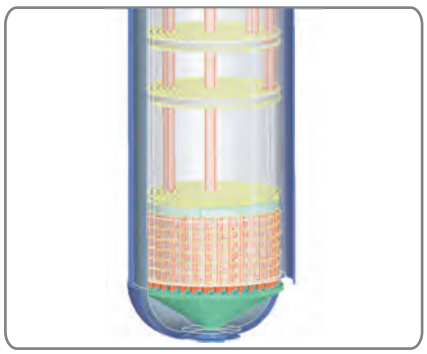
The above, along with the user-friendly solution scenario definition leads to the output of a ready to solve model. Apart from the formats of the solvers for which complete preprocessing decks exist, i.e. NASTRAN, LS-DYNA, PAM-CRASH, RADIOSS, PERMAS, Abaqus/Standard, Abaqus/Explicit and ANSYS, numerous other file formats are supported, for structural, CFD as well as other solvers.

Morphing

One of the most advanced tools developed to meet the needs for fast model modifications is the Morphing Tool. It expands the reusability of existing models by allowing the versatile procession of alternative variations. Morphing operations are performed on FE or geometry models. Applications include fields like optimization processes and sensitivity analysis.

Tools

The core preprocessing functionality is enhanced with a substantial number of other advanced tools that allow the user to complete specialized tasks without having to leave the software environment. Those tools are the Cross Section Analysis Tool, the Volume Traps Tool and the Fuel Tank Analysis Tool.



Optimization

Numerous software features, including the Morphing Tool, the scripting and the process automation, can be combined to offer versatile coupling with optimization codes.

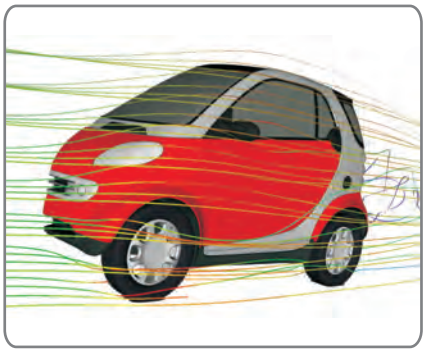
μETA is a thriving multi-purpose post-processor that meets diverging needs from various CAE disciplines. Combining its highly sophisticated nature, the user friendly interface, and its automation capabilities, μETA facilitates effective and thorough post-processing. Additionally, its impressive performance, its innovative features, and the capabilities of interaction between animations, plots, videos, reports, and other objects, makes μETA a complete solution for fast and advanced post-processing for every CAE discipline.

Supported results

μETA provides a broad range of functionality that can successfully address even the most demanding post processing requirements for structural and CFD analyses. It supports results from all popular solvers used in structural analysis, as well as CFD results from Fluent and OpenFOAM. Data from neutral format files, such as ASCII column, Universal and ASCII Patran, are also supported.

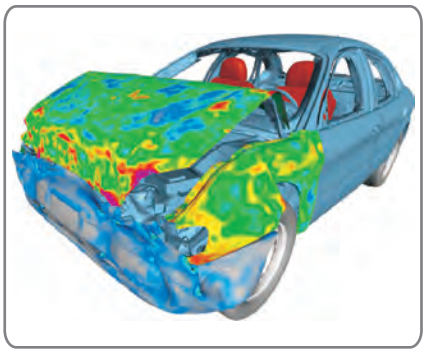
User interface

A flexible and fully customizable interface provides the option to dock existing toolbars anywhere on the working area and even create new ones, suiting diverse needs. Post-processing work is accelerated significantly through a multiple window environment with window dependent model attributes and smart functionality for the fast handling of entities' visibility.



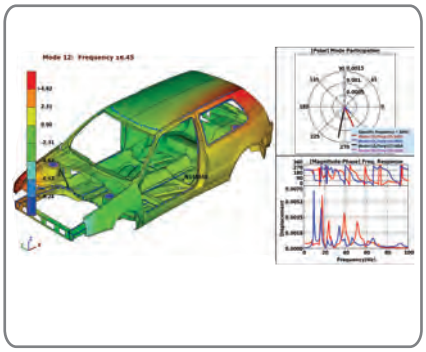
3D field post-processing

The top quality graphics, the high performance and the efficient handling of large multiple models and data form a productive working environment. Model management, conducted on entity, property or material level, is greatly assisted by a tree-structure hierarchy group tool that reflects the ANSA Part Manager hierarchy and provides connections management capabilities amongst others.



2D plot post-processing

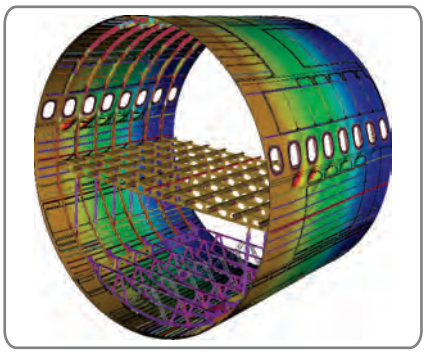
A complete, comprehensive graph tool is integrated within μETA without the need to start a separate program. As many 2D plot windows as necessary can be opened, each one hosting multiple different plots. This tool supports direct reading of time history results deriving from all supported solvers, as well as PAM-VIEW ASCII files, ISO DATA files or common column ASCII files. Its unparalleled speed in interactive handling of large number of curves, as well as its extensive interoperability with the 3D field module, boosts productivity and simplifies parallel 3D and 2D post-processing. Information can be passed from one module to the other and curve data can be directly associated to the 3D geometry. Mathematical operations and a broad range of built-in or custom functions can be applied on curves or a field deck.



Original model geometry courtesy of Volkswagen AG

Calculations on field results

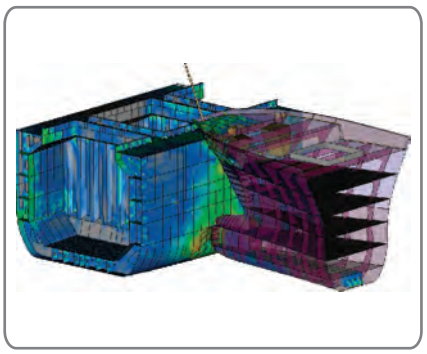
New stress fields can be calculated in μETA as a linear combination of existing stress fields or through any mathematical operations applied on loaded results. Forces on any section can be calculated from original results stemming from NASTRAN, Abaqus, LS-DYNA or PAM-CRASH. NVH



analysis is accelerated by the fast calculation of 3D and 2D structural and acoustic responses, the generation of structure-fluid coupling data and the modal correlation capabilities (MAC calculation). Hybrid modeling is also supported for NVH analysis framed by the creation of modal models and an FRF assembly tool that also features an embedded genetic optimizer for the optimization of connectors' properties.

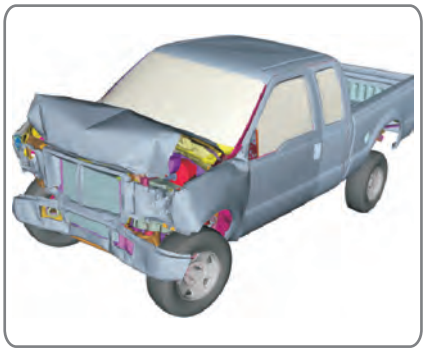
Advanced filtering and communication of results

µETA offers extended interactive identification and filtering capabilities on part and element level. Areas of interest and the corresponding Maximum / Minimum values can be marked and traced easily through the run-time annotation tool and the fast creation of iso-contour lines and surfaces. Advanced filtering is available within several µETA tools, with which even extreme cases of entities selection and information extraction can be realized with a single click based on combined queries. An overview of hot-spot results, as well as spreadsheet properties is achieved through multiple statistics tables which also provide an efficient means for fast comparison of results.



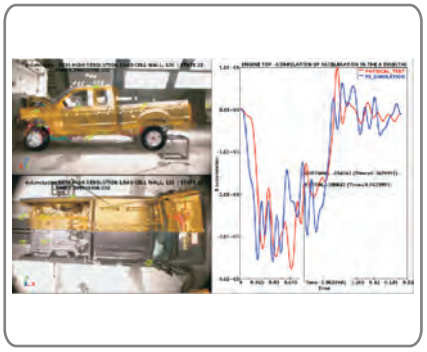
Output options

Geometry and results data can be saved and appended in the native binary format of µETA. The content of these files can also be viewed with a freely available viewer. Geometry can be output in NASTRAN bulk data or PAM CRASH input file format. Data related to identified entities as well as curve data can be output in ASCII text, while data from the spreadsheet-form tools can be output both in ASCII and HTML format. µETA also provides image saving for tif, jpg, ppm, png, bmp, PostScript, encapsulated PostScript, vrml, rgb and gif formats and video recording for avi, mpeg and gif formats.



Automation

µETA provides advanced capabilities for process automation through scripts in the same scripting language used in ANSA, custom user toolbars and session files, all of which can be created easily and fast through intuitive embedded editors. Based on the integration of these means, specific tools for the automated creation of reports for several crash safety regulations (pedestrian, IIHS, etc) and the conductance of bore distortion analysis have been developed and are distributed freely with µETA. The automation capabilities also contribute to the easy and unhindered coupling of µETA with external optimizers. A simple-to-use toolbar, that streamlines the coupling, is provided with the standard package.

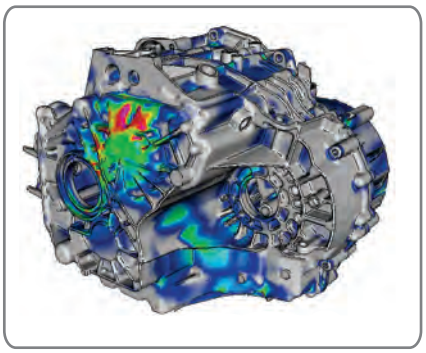


Other tools

Other tools such as the multiple clipping-plane-cuts with its advanced functionality, the explode feature with the numerous options for parts and models and the section force calculator provide the means for a closer insight into the results. Furthermore, a camera tool ensures the accurate management of the perspective view of the model.

Correlation studies using videos & images

A number of specially developed embedded tools are available to significantly simplify correlation studies between simulation results and physical test data. Using these tools, the exact and effortless matching of the model's perspective view with that of an image can be achieved and the synchronisation of the animation simulation with a test video can be set. Additionally, features can be traced on videos and the tracking results can be plotted automatically in a 2D plot.



Courtesy of Volkswagen AG

Reporting

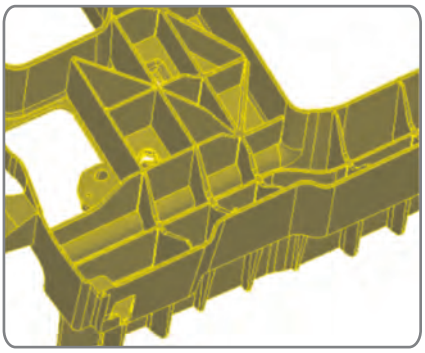
Using the embedded report composer, template-based reports in HTML, PostScript or MS Office PowerPoint .pptx format can be generated in a direct way. The reports can be created interactively, taking advantage of the complete text editor in addition to the interoperability between the different tools of µETA, or automatically through the use of session files.

ANSA
pioneering
software systems

quality and performance in automatic mesh generation

courtesy of Dr. Ing h.c.F. Porsche AG

ANSA, combining the automatic geometry healing with the automated middle skin extraction and the Batch Meshing, offers the most efficient creation of meshed models. The resulting shell and volume elements models easily meet the meshing requirements and quality criteria in less than 75% of the time needed by other software.

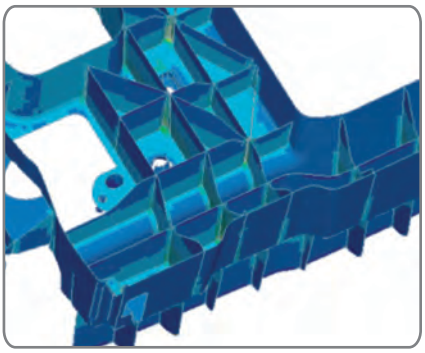


Geometry handling and area idealization

Geometric entities are easily handled and manipulated by numerous integrated tools. Geometrical errors are automatically identified, isolated and fixed, while depending on the mesh requirements, the geometry can automatically be simplified.

Advanced fully automated middle surface extraction

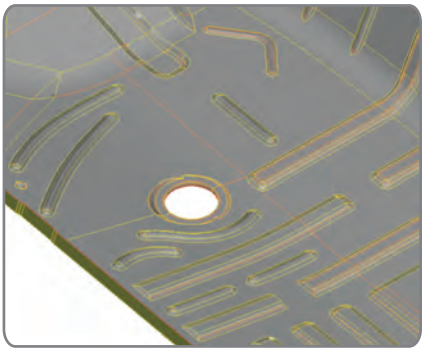
ANSA offers a variety of functions allowing a fully automated extraction of the middle surface for stamped and cast parts of any complexity. Without the need for user intervention, starting from the solid description, high quality mesh is created at the neutral fiber, while nodal thickness is assigned.



Meshing algorithms

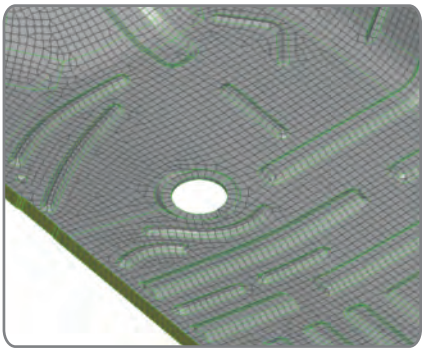
Following the mesh area idealization, geometry can be meshed according to modeling requirements by cutting edge fully integrated surface mesh algorithms. These algorithms are uniquely created for:

- Sheet metal components for all analysis types (crash, NVH, durability, etc)
- CFD applications, with resolution adapted to local surface curvature and flexible size boxes for the automatic refinement of specific regions
- Surface Wrapping of solid description, capturing sharp edges, with curvarture dependent mesh density
- Shell and Solid meshing for structural analysis
- Tetra meshing of high quality
- Boundary layers element deployment
- Hexahedral dominant mesh
- Pure hexahedral meshing based on multi block structures
- Polyhedral mesh
- Acoustic cavity mesh

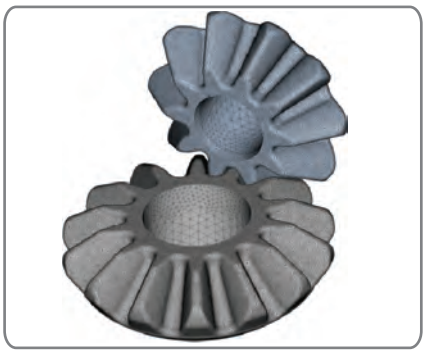


Mesh Quality control

The quality of the mesh is easily evaluated using the numerous available quality criteria. Violating elements are isolated and colored according to criteria type or value and automatically corrected. Moreover, automated mesh information reports and quality statistics are provided.

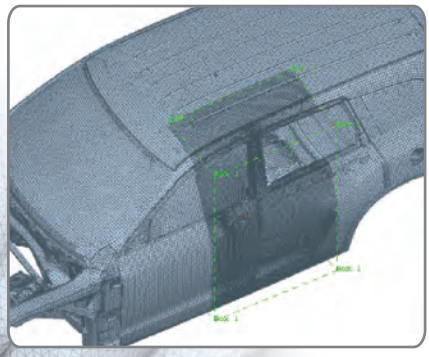


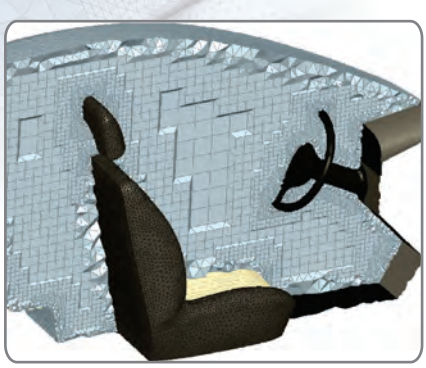
Automatic feature and mesh handling



A number of automatic functions are available for mesh manipulation and improvement. ANSA's responsive graphical user interface guides the user in creating a mesh meeting all requirements for each analysis type. In addition to the numerous proprietary mesh generation and reconstruction algorithms, ANSA provides:

- Automatic feature recognition and defeaturing (selection of defeaturing level, logos and emboss removal, identify and respect rib edges in solids)
- Hole / tube treatment (identify bolt holes, fill holes, assign nodes, create zones, etc.)
- Fillet treatment (sharpen, split, assign element rows depending on size and curvarture, length control along fillet according to aspect or given value)
- Chamfer treatment
- Flange treatment (identify flanges and apply treatment in both shell and solid meshed parts)
- Local refinement or coarsening of mesh





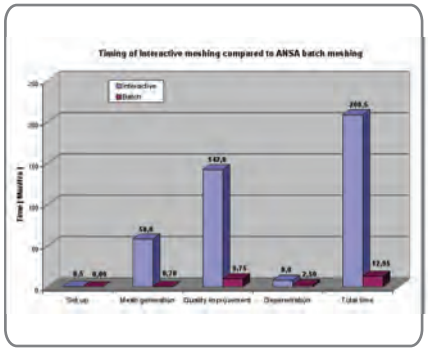
Hexa dominant mesh

Features

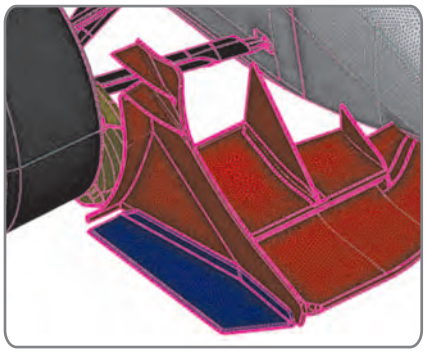
- Controllable meshing
- Predictable results
- Multidisciplinary process
- Shell & volume meshing
- Quality improvement
- Repeatability
- Local refinement / coarsening
- Reporting
- Advanced automated mesh area de-featuring
- Controlled washers
- Element rows on fillets and flanges
- Logos and emboss removal
- Hexa Interior
- CFD Mesh
- Fillets Split or Sharp
- Tubes treatment
- Tetra Mesh
- Solid structural mesh
- Mapped mesh on fillets and tubes
- Cooperation with ANSA Connection Manager
- Intuitive mesh parameters generator

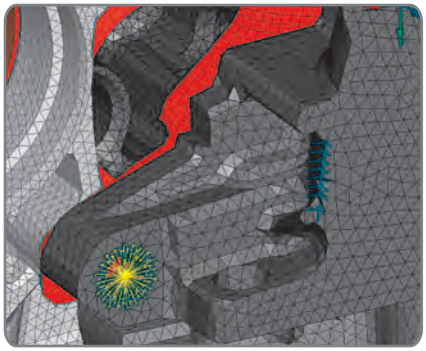
Benefits

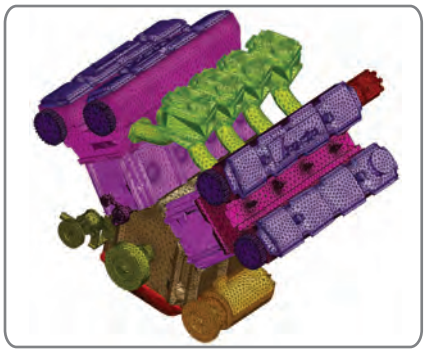
- Significant meshing time reduction and consequent reduction of the overall modeling time
- High mesh quality, regardless of user's experience
- Decrease of human error factor
- Repeatability of meshing operations
- Process efficiency enhancement through the integration of the batch meshing process with the rest of the pre-processing functionality, within the same environment
- Adaptability to alternative technological cultures of different processes, followed by different CAE environments and disciplines

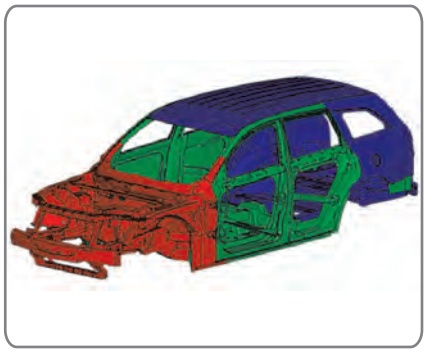


The ANSA Batch Meshing Manager is a powerful tool for the versatile, controllable and predictable high quality meshing with shell (surface, CFD, wrapping of 3D parts), or volume (solid components, CFD, HEXA interior) elements.
Being completely integrated into ANSA, it is capable of operating through a GUI as well as non-GUI batch process mode.







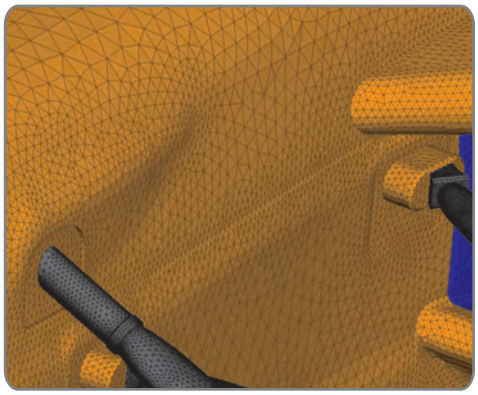


The concept of the batch meshing scenario

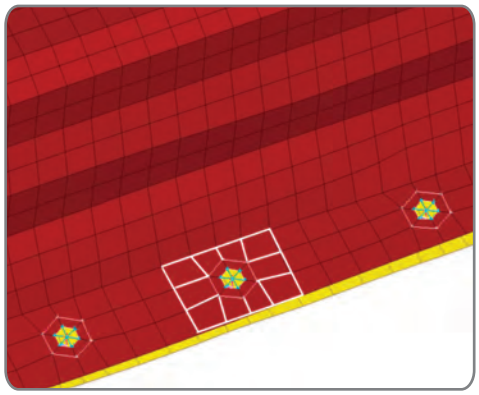
The principal concept of the ANSA Batch Meshing Manager is based on the fact that usually, each area of the model has to be meshed with different meshing parameters, depending on the discipline and the simulation scenario.
In this manner, a Batch Meshing Scenario is partitioned into Batch Meshing Sessions, which are collections of model parts that are corresponding to the areas to be treated with different settings. These areas can be identified according to the parts' location, weight, external dimensions. The settings of each session can be tuned right from the early stage, in order to provide control over four main factors:

- the shell & volume mesh properties,
- the features treatment rules,
- the quality improvement settings and
- the mesh quality criteria

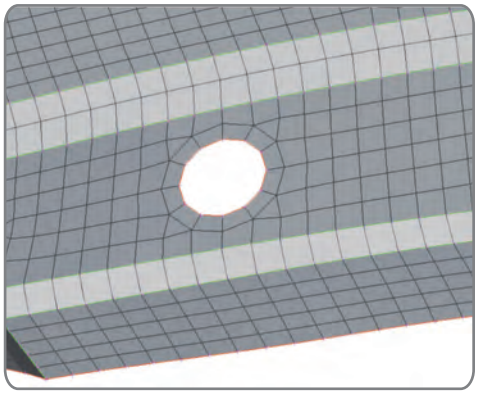
The ANSA Batch Meshing Manager allows the detailed definition of the Meshing Scenarios. All required operations can be achieved through this interface, including the monitoring of the process execution status and the reviewing of the results report. Additionally, the set up and the execution of meshing sessions can be a straight forward GUI driven procedure, while previously set up and saved Meshing Scenarios can also be run and re-run in a no-GUI environment powered by the ANSA scripting language. It is notable that ANSA Batch Meshing Manager not only generates mesh according to specifications, but simultaneously improves the quality of the mesh under process, so that the result meets the quality criteria that are declared in the template. Consequently, the resulting mesh is of higher quality compared to interactively generated mesh and less time is required for any further inspection and correction. The optional further interactive improvement of the batch meshing result may be achieved through the core ANSA functionality. This allows for the selective modification of mesh in critical areas and therefore achieving its improvement in a controllable manner.



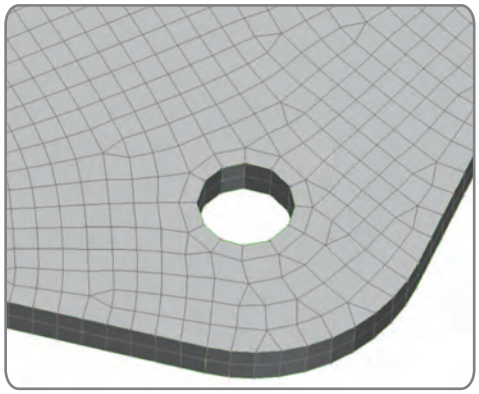
CFD meshing



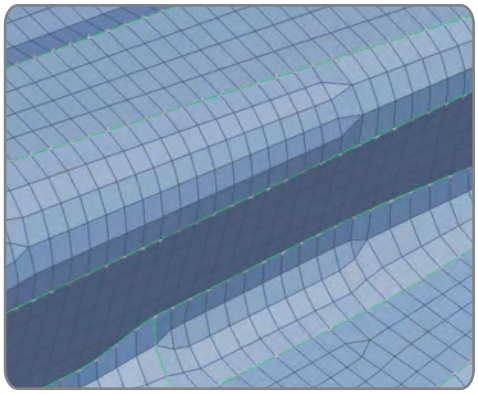
Spotweld mesh patterns



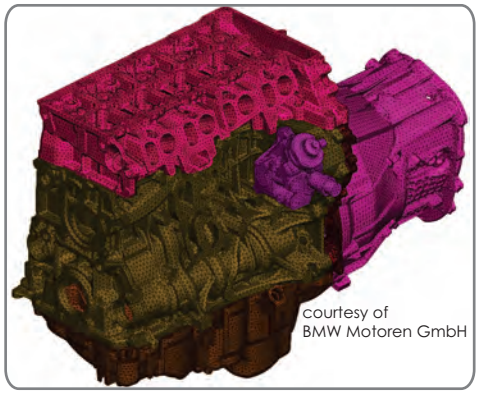
Flange and hole treatment



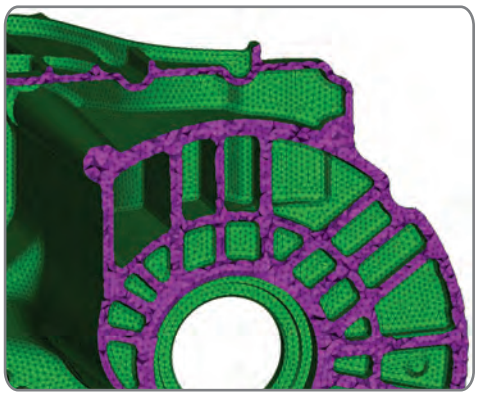
Tube treatment



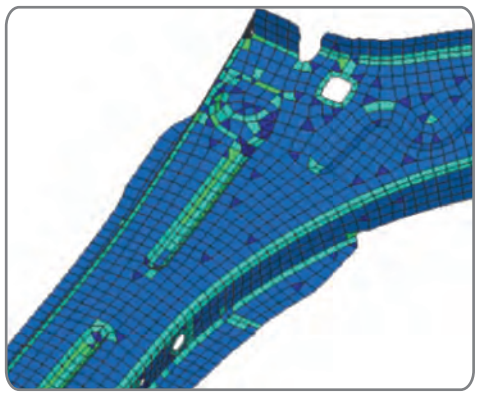
Automatic fillet treatment



Solid structural mesh



Mapped mesh upon request

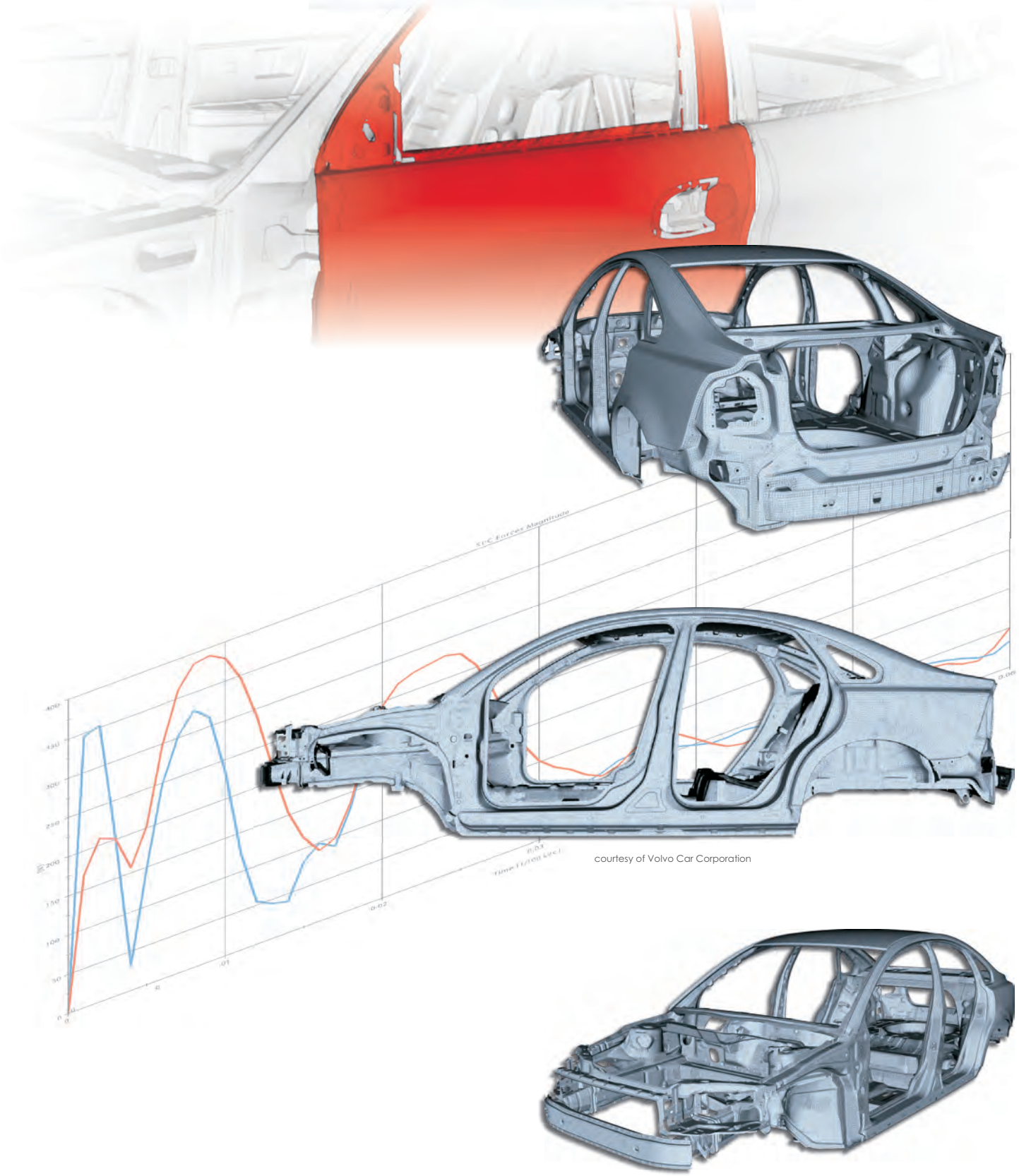


Quality criteria graph

ANSA`s Batch Meshing Manager drastically reduces the human error factor in addition to minimizing meshing time for model completion, thereby achieving a high quality mesh regardless of user's experience.

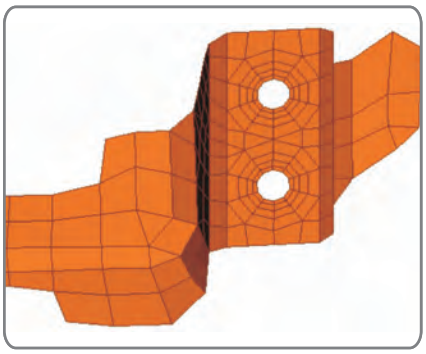
ANSA
µETA
PostProcessor
pioneering
software systems

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



courtesy of Volvo Car Corporation

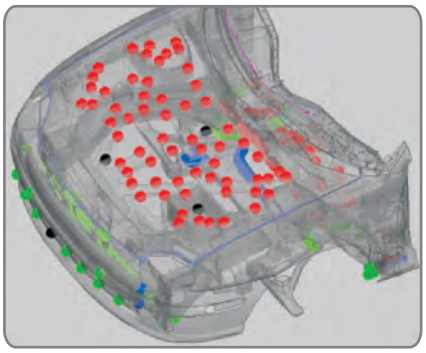
ANSA is the only pre-processor capable of increasing the efficiency of the detailed FE-modeling for Durability, as it provides a variety of features suitable for demanding engineering tasks that meet the requirements of today's industry. It efficiently supports popular codes, such as Abaqus/Standard, 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

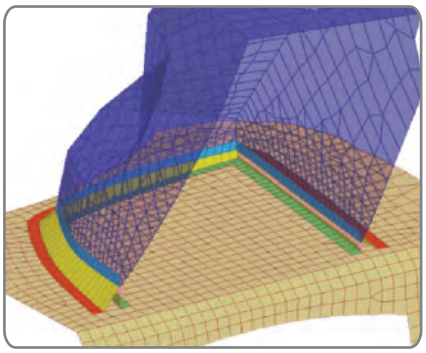
ANSA offers a wide range of meshing options and features adapted to durability specifications.
Proper local refinements in crucial areas - a common task of analysts - are achieved in a way such as to meet several durability requirements:

- 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

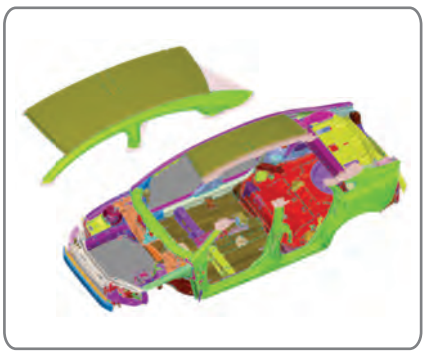
ANSA provides the capability of a full overview of the model, its components, and their changes. These may be visualized via multiple navigation methods, such as per parts, per include file, or per keyword. Moreover, the model part representations can be automatically controlled, even in case of multi-instantiated 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.



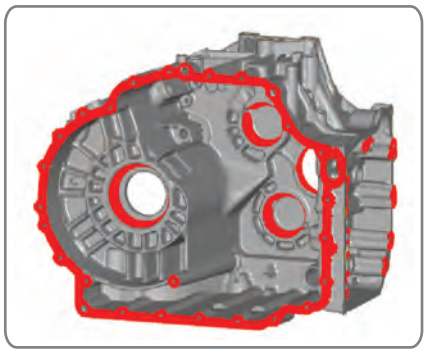
Weldings modeling

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 or seam lines
- Concrete position of connection representation
- Suitable element definition with controllable quality
- Detection & enhancement of unsuitable connections information

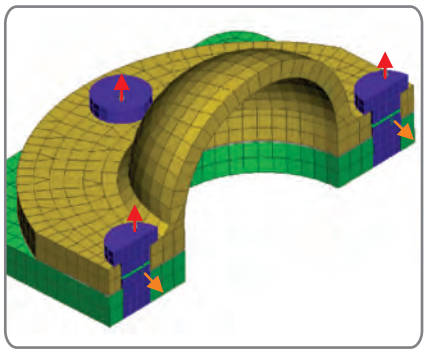


Model cut – Abaqus substructure



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 Abaqus Substructure 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.

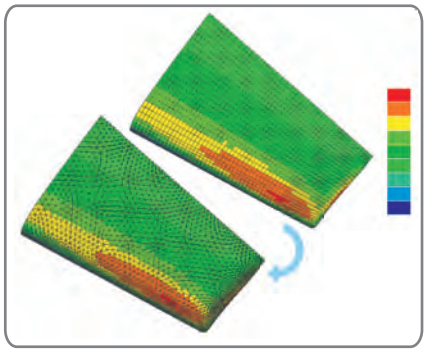


Contacts management

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.

Pre-tention Assistant

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

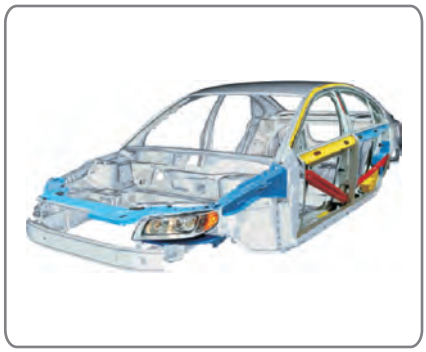


Features

- Interoperable decks
- Process automation
- Model assembly
- Include-files handling
- Mastering of entities' IDs
- Weldings modeling
- FEMFAT welds modeling
- Laminates Tool
- Positioning & Kinematics Tool
- Contacts detection
- Mesh refinement
- HEXA block meshing
- Arbitrary cross sesction calculator
- NASTRAN SOL 600 set up
- Substructuring
- Results mapping
- Quality validation & fixes
- Pre-tention assistant
- Analysis manager

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



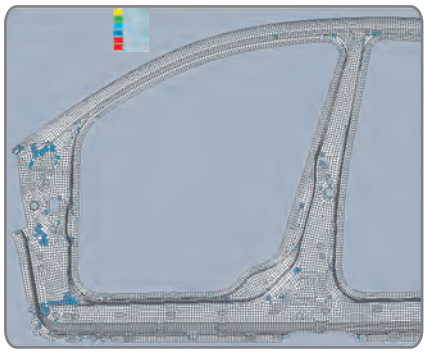
Courtesy of Volvo Car Corporation

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.

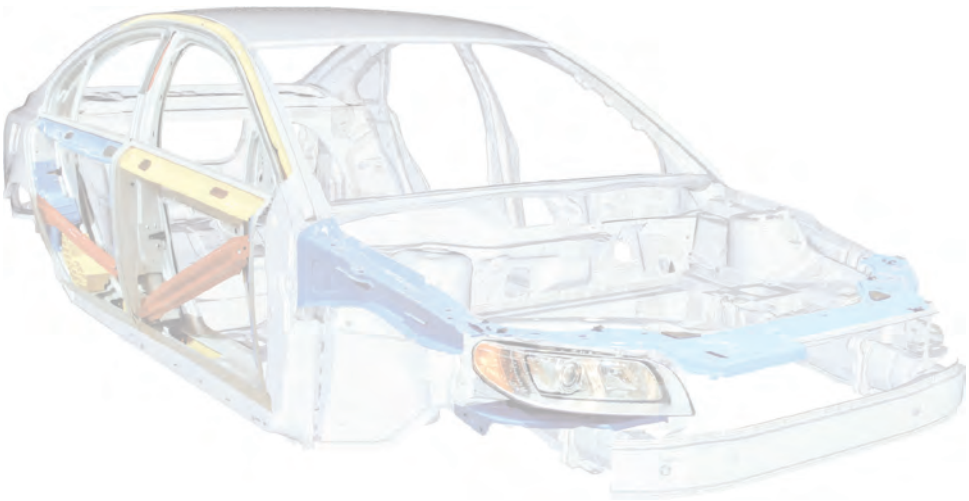
Task manager – Load cases definition

The Task manager module organizes the sequence of steps and actions that caputure 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.

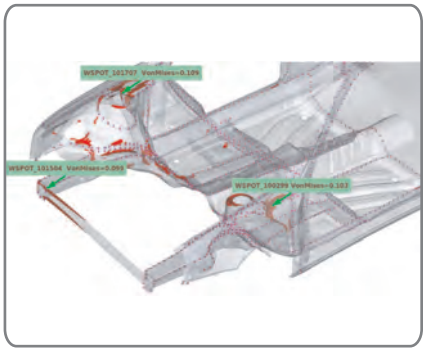


Quality checks & improvements

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.

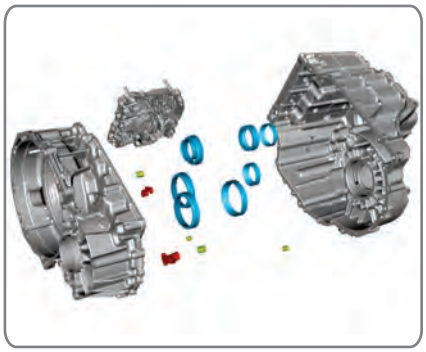


μETA is a highly sophisticated yet user friendly post-processor. Through its continuous development it has succeeded in meeting the ever increasing demand for efficient and thorough post-processing. μETA supports results from all popular static and fatigue analysis solvers, as well as real-life test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

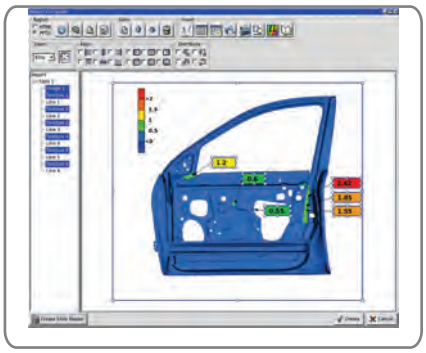
Smart and intuitive main functionality with numerous options succeeds in managing and manipulating the model requiring minimum time and effort. Results from contact analyses are conveniently displayed and efficiently handled through a list. A complete sets tool advances model handling capabilities to 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

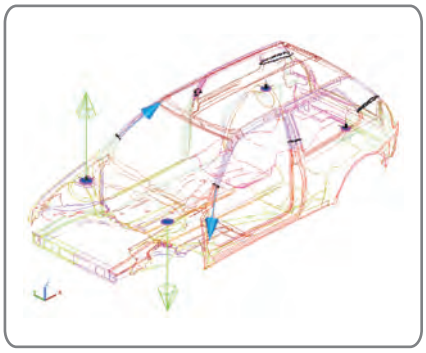
μETA has versatile filter capabilities incorporated in many of its tools. Filters can be used alone or combined based on results, names, position, regions and many other criteria to identify, isolate or pin-point areas of interest (hot spots). An overview of these areas' results is achieved through statistics 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 μETA, modified and previewed in slide show mode.



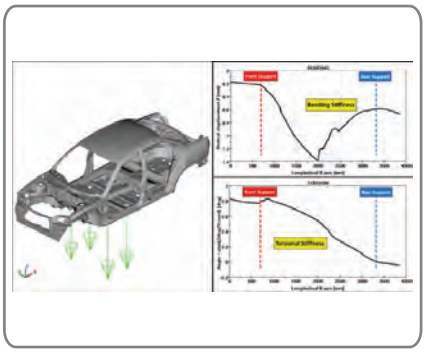
Results calculation

Addressing specific needs of durability and fatigue analysis, μETA features a calculator for the linear combination of results deriving from other loadcases. This calculator is accessed through a user friendly interface and can also be fed with load history curves to provide the history of the linearly combined stress field. 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.



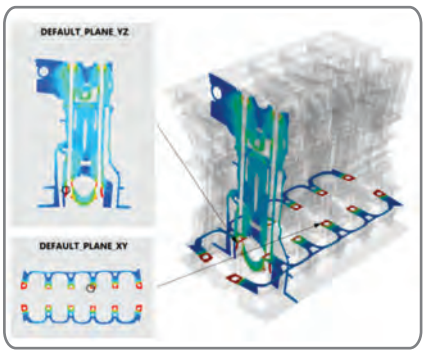
2Dplots

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). In addition, model data as well as other post-processing information is stored in μETA native binary files which can also be viewed with a license-free viewer. Among other outputs, deformed geometry, cross sections and iso-contours can be exported in solver's format.

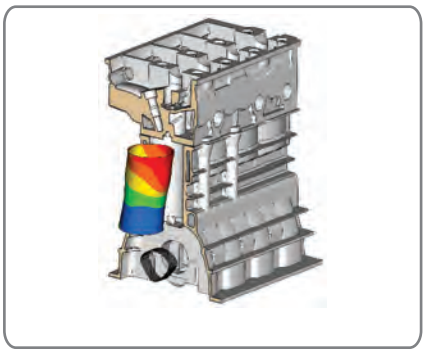


Courtesy of Volkswagen AG and Forschungsvereinigung Automobiltechnik

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, thus boosting productivity and assuring the quality of final report. Furthermore, the coupling of μETA 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.

Finally, the user can benefit from the automation capabilities and perform bore distortion analysis using another toolbar. Bore rings are defined through an intuitive GUI and distortion fourier orders are calculated along the bore length. The cylindrical deformations are displayed in polar plots considering only selected fourier orders.



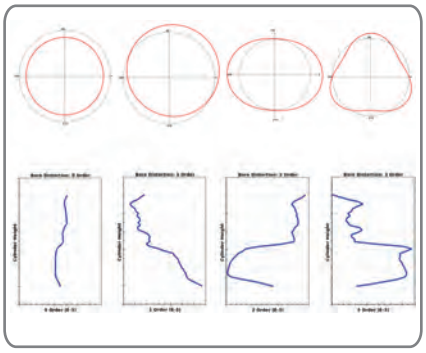
Courtesy of Volkswagen AG and Forschungsvereinigung Automobiltechnik

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

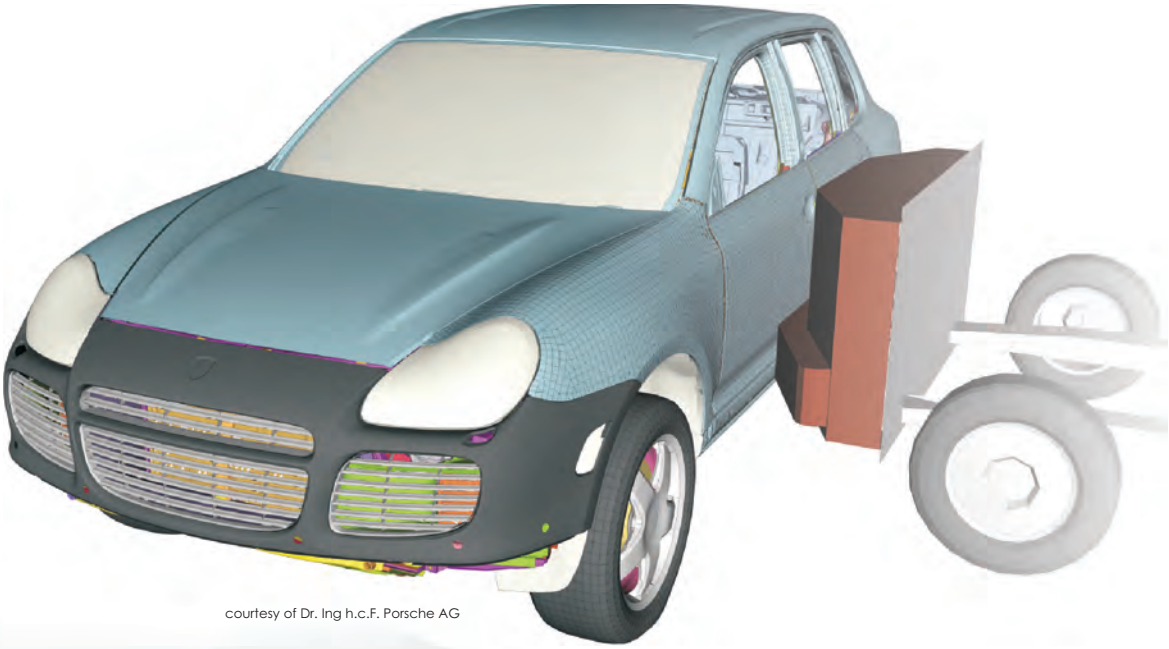
Benefits

- 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

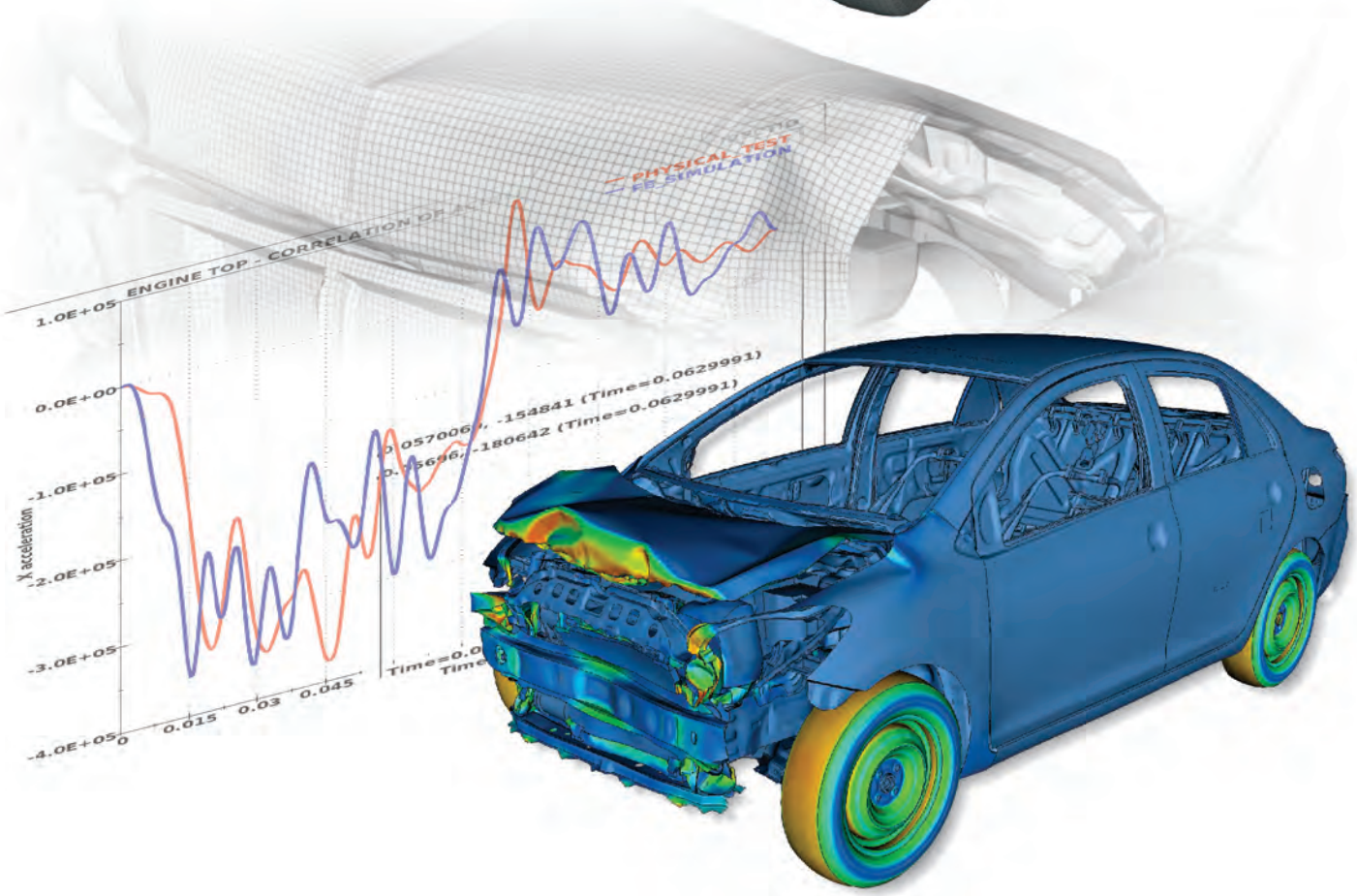


ANSA
µETA
PostProcessor
pioneering
software systems

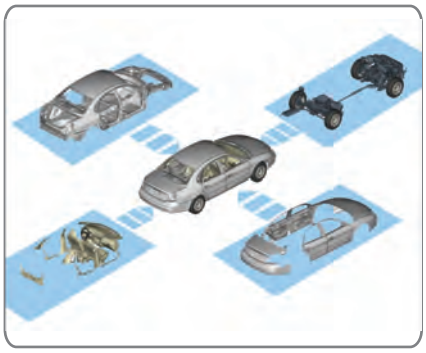
the standard in crash & safety pre- & post processing



courtesy of Dr. Ing h.c.F. Porsche AG

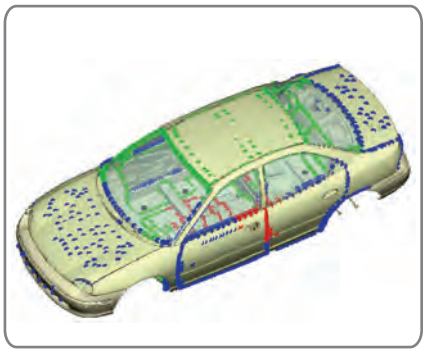


ANSA is the Industry standard solution for demanding crash & safety pre-processing. The software's powerful functionality allows users to efficiently perform the required tasks in less time and at lower cost. ANSA supports all common solver keywords used in modern modeling techniques of crash & safety solvers. The supported solvers are ABAQUS/Explicit, LS-DYNA, PAM-CRASH/SAFE and RADIOSS. The innovative ANSA concept of interoperable decks allows a model to be easily converted from one solver input deck to another, providing superb flexibility. Process automation and data management are an area extensively supported by the ANSA Task Manager and ANSA Data Manager. These tools allow for fast, repeatable and robust model build-up and loadcase definition processes. Additional tools such as morphing and optimization coupling, leverage ANSA to a multipurpose software package that meets the needs of even the most demanding users.



Include files Configurator

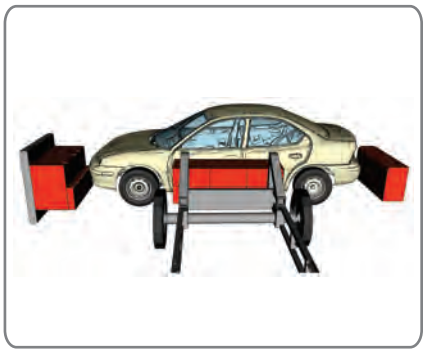
The basic concept of the Include files Configurator is the generation of "ready-to-run" main decks using existing includes. Different configurations of the simulation model can be defined in model as well as in load case level. These configurations can be exported at once without the necessity of reading the include files into ANSA. In cooperation with the ANSA Data Management features, include versions, and representation management are achieved, building a unique environment for the build-up of simulation models.



Weldings modeling

Numerous semi and fully automatic tools are available for the creation of modeling welds, based on definitions made within the software or on information imported by a PDM system.
The Connections Manager, allows multiple connection type definitions to be completed in a single step, thereby achieving:

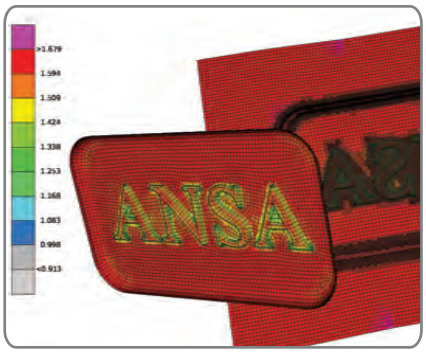
- Wide variety of weld definition types for spot welds, adhesives, bolts or seam lines
- Flexible re-definition of connection elements, to serve different modeling purposes
- Detection & improvement of improper connection information
- Configuration files and templates allow the standardization of connections creation, ensuring repeatability and enforcing model robustness



Load case definition

ANSA provides wizard driven assistants for complicated tasks, requiring minimum effort, such as:

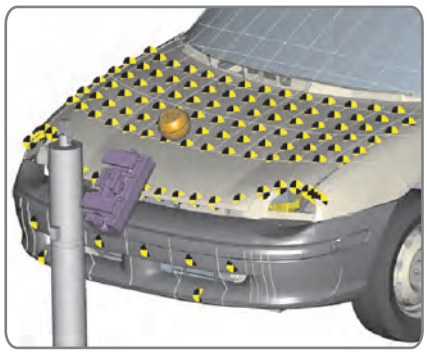
- Positioning of impactors and roads
- Keyword definitions like joints, rigid bodies or output requests (time history, section forces)
- Advanced mass trimming
- Mass balancing to achieve target weight and COG and many more



Results mapping

The usage of initial conditions during a crush simulation requires high quality mapping of results taken from other calculations, such as sheet metal forming or draping of laminates. For this purpose, the Results Mapper maps thickness, pressure, stress, plastic strain or material

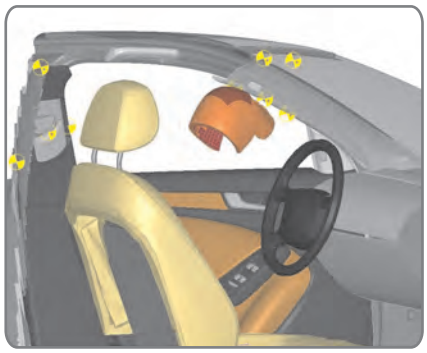
orientation information, offering:
- Supported solver formats: NASTRAN, ABAQUS, LSDYNA, RADIOSS, PERMAS, ANSYS, PAMCRASH, PAMSTAMP, FIBERSIM, SIMULAYT
- Variety of interpolation methods
- Full-automatic positioning tool available, in order to position the source part on to the target part, achieving a "best-fit"
- Semi-automatic tools, to preview and correct the source part positioning
- Validation of the mapped results
- Capabilities to specify user defined results readers, interpolation and validation methods
- Fully operable in batch mode
- Capability to handle results in different unit system



Occupant & pedestrian safety

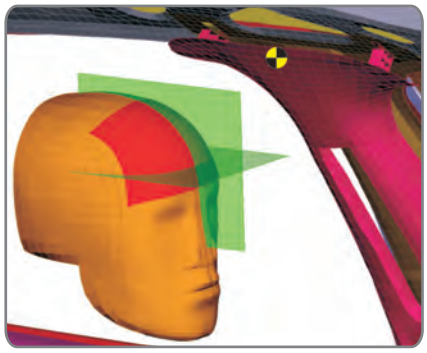
ANSA, in alliance with the suppliers of Crash Test Dummy models, accurately supports the commonly used dummy models and structure trees. ANSA offers the necessary functionality for:

- Positioning and articulation of Dummies, respecting the joints' rotation stop angles
- Creation of Dummies' structure, in case of absence of structure and positioning data
- Intuitive restraining for seatbelts systems definition and fitting
- Dummy-seat depenetration
- Coupling of dummy and seat allows the combined movement of both



Powerful and innovative functionality is provided for:

- Pedestrian safety tools (EuroNCAP v6.0, EuroNCAP Grid Proposal 2010, EU Phase 1 & 2, JNCAP, TRIAS 63): calculation of reference lines, critical impact points and headform / legform positioning
- Interior impact protection tools:
  - FMVSS 201U calculation of target points and automatic (contact based algorithm) positioning of the FMH
  - FMVSS226 ejection mitigation
  - Instrument panel impact protection tools (FMVSS 201 / ECE-R21)
  - Seat impact (ECER17, ECCR21, FMVSS202A) calculation of zones, positioning of headform

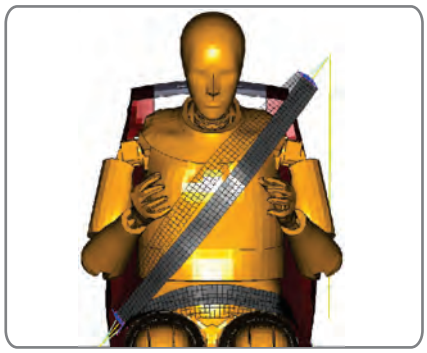


Features

Complete modeling for:
- ABAQUS/Explicit
- LS-DYNA
- PAM-CRASH
- RADIOSS
- Interoperable decks
- Process automation
- Model assembly
- Include-files handling
- Mastering of entities IDs
- Mass trimming
- Kinematics tool
- Dummy positioning & restraining
- Pedestrian & occupant safety
- Quality validation & fixes

Benefits

- A complete pre processing environment that offers a portfolio of features that covers the whole Crash and Safety area
- Intelligent interface guides even the non expert user to the fast and smooth application of all functions
- Reduced pre-processing time

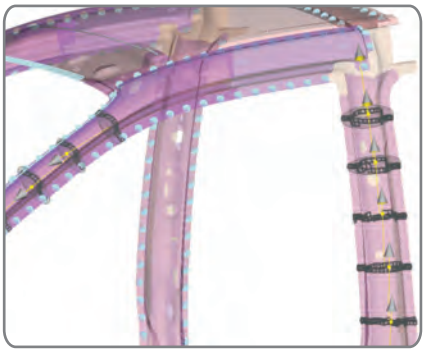


Seat model courtesy of BMW AG

Positioning & Kinematics tool

The Kinetics module tool is an implicit multi-body solver used to move complex kinematic mechanisms. Application examples are movements of suspensions, seat and dummies, convertible roofs, hoods and many more. The main features of the tools are:

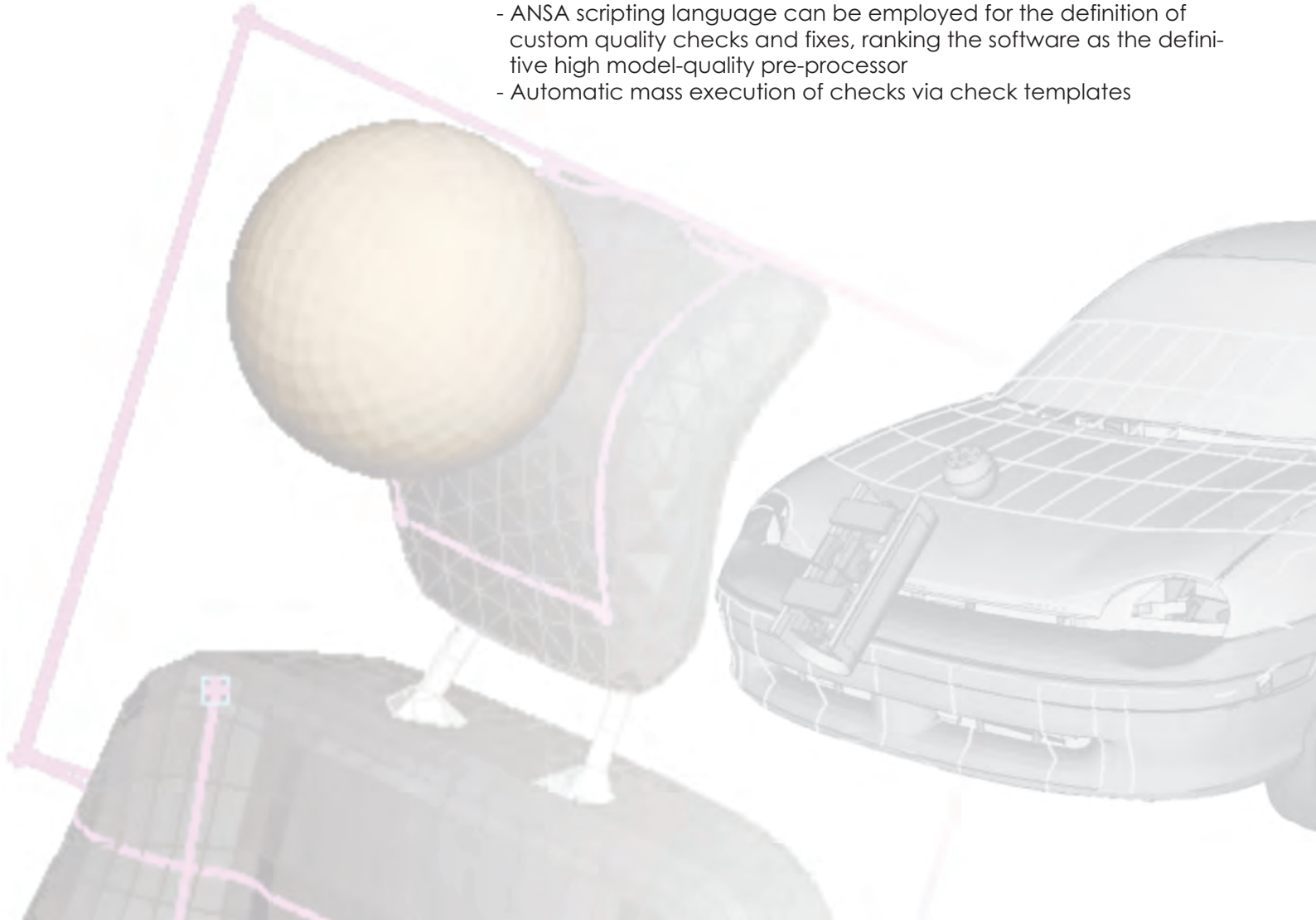
- Kinematic model definition in a single step
- Positioning of systems like seats, dummies, convertible roof systems, suspension and complete steering mechanisms without the use of an external solver
- Creation of saved positions, in order to retrieve any applied configuration of the model
- Transformation information export, without writing the whole FE-model



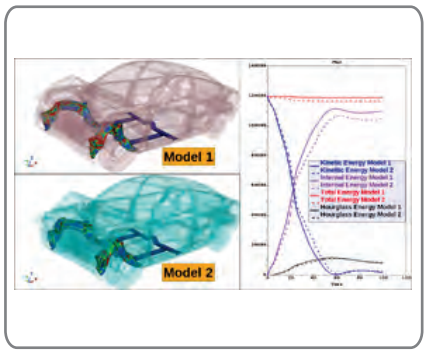
Quality checks & improvements

A variety of checks are available in ANSA in order to verify the integrity of the model.

- Mesh quality checks and automatic fixing algorithms are provided, based on crash solvers' quality criteria and thresholds
- Intersections and penetrations are located and removed
- Incorrect Contacts & Tie definitions or unconnected parts are identified and corrected
- Detailed mass information is calculated and reported, according to the mass scaling parameter, always in full compliance with the solver
- ANSA scripting language can be employed for the definition of custom quality checks and fixes, ranking the software as the definitive high model-quality pre-processor
- Automatic mass execution of checks via check templates

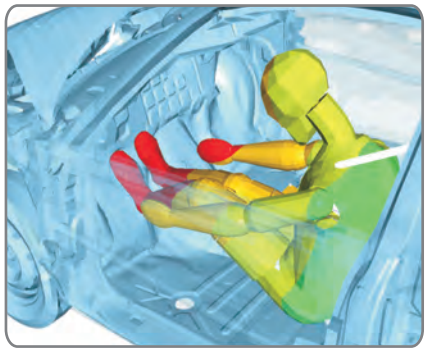


µETA stands up to the modern crash-analysis challenges, such as the increasing model size and memory and numerous model comparisons, thereby enabling easy and fast correlation studies with real videos and procedure automation.
µETA features a complete integrated graph tool for 2D post processing which exhibits unparalleled speed in handling a large number of curves. Time history results from most popular crash solvers are supported while parallel 3D and 2D post processing can simply be conducted based on the broad range of built-in functions as well as on the extensive interoperability between 3D and 2D modules.
The comparison of large models is greatly facilitated by efficient memory usage and by the capability of limiting loaded results to only a few parts of a model. Models handling, 2Dplot tool, cut planes, calculation of models differences and multiple statistics windows exhibit functionality that simplifies model comparison so as to be performed in just a few clicks.



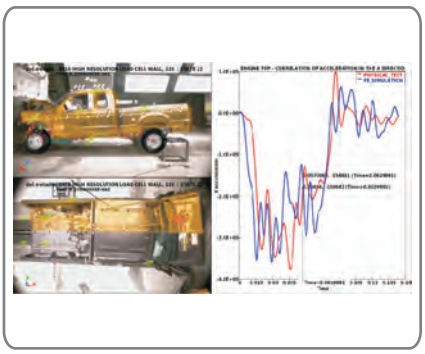
Model management

µETA addresses the need for simple and rapid view control of models, consisting of numerous parts and groups, through its intuitive one-click functionality. This capability is further augmented to meet extreme handling requirements through window-dependent model attributes and the definition of multiple undeformed states which also allow for simultaneous display of a model at different time steps (film strip display). Furthermore, the model assembly can be reflected in a tree form inside groups tool. Weldspots are realized as groups and their respective connectivity is retained.



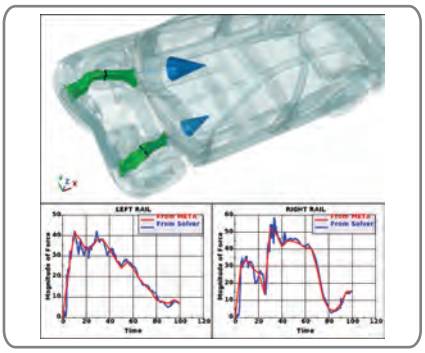
Results extraction & calculations

Distances between nodes, elements, or even parts, planes and groups can be rapidly identified and updated in real time following the animation. Proximity between groups can be displayed as contour plots while collided elements between groups can be isolated.
Displacements, velocities and accelerations can be transformed both in 3D and 2D level relative to a moving coordinate system providing intrusion results that can be displayed either as contour/vector field or as graphs.
The analysis of sections is elevated through the cut planes tool and the use of run time annotations on planes. Using the section force calculator, results on cross sections which were not defined in the solver's input file, can be obtained, therefore avoiding rerunning the solution.



Correlation studies using videos & images

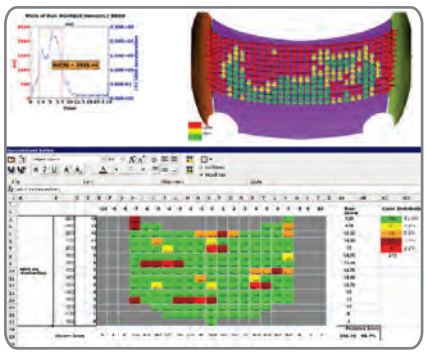
The virtual camera can simulate onboard cameras, while the image matching and video synchronization turn into easy tasks with the aid of embedded tools. Graphs of displacements and angles between traced features on test videos generate useful information also to be used for correlation with simulation results.



Project files and free viewer

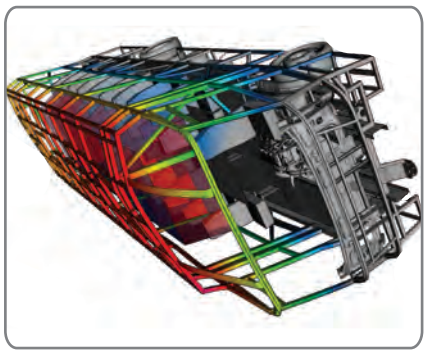
µETA Projects are native binary files for compacting and storing necessary post-processing data, such as model geometry and results, graphs, videos, reports. Project files can also be accessed using µETA Viewer, the freely distributable reduced version of µETA. This can be used either as a standalone tool or as a plug-in of web browsers or MS Office applications.

Reporting & data communication

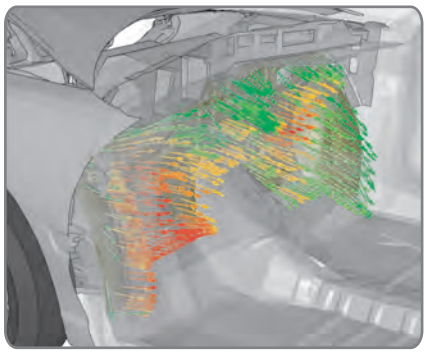


Reports in html, PostScript or MS Office PowerPoint .pptx format can either be created interactively through an intuitive interface or through the use of scripts or sessions, so that the analyst can receive immediately evaluable results from the automatic procedures. An embedded spreadsheet editor allows for the input/output and further processing of data. The interactive run-time creation of reports becomes a very easy task through dragging/dropping of images, spreadsheets and other data to the report composer. pptx reports can also be input in µETA and previewed in Slide Show mode. Deformed geometry as well as cross-sections and iso-contours can be output in solver's file format.

Crash safety & process automation



A simple-to-use toolbar designer ensures the fast creation of user toolbars for the standardization of crash analysis procedures. The analyst can take full advantage of the parameterized sessions and unique scripting capabilities, which can lead to automated streamlined processes regarding the extraction of specific results or model comparison.
Crash Safety Analyses can be realized completely inside µETA. A great variety of tools, such as colored annotations and the embedded 2D plot tool with the internal calculation of Crash Analysis Criteria provide the means for such analyses, while the use of µETA scripting language can drive to full process automation.
These automation capabilities also enable the effortless coupling of µETA to external optimizers which can be readily realized through the use of a provided toolbar.

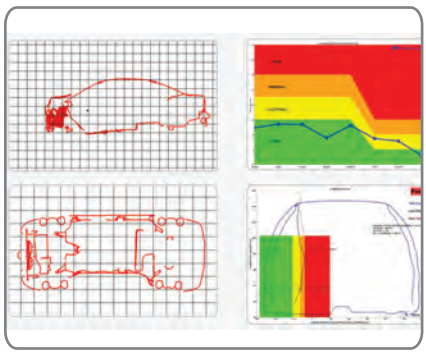


Features

- 3D & 2D post-processing
- Numerous interfaces
- Process automation
- Scripting
- Parameterized sessions
- Intrusions calculation
- Section Forces calculation
- Crash criteria
- Video & image correlation
- Annotations
- Reporting
- Native database
- Free viewer

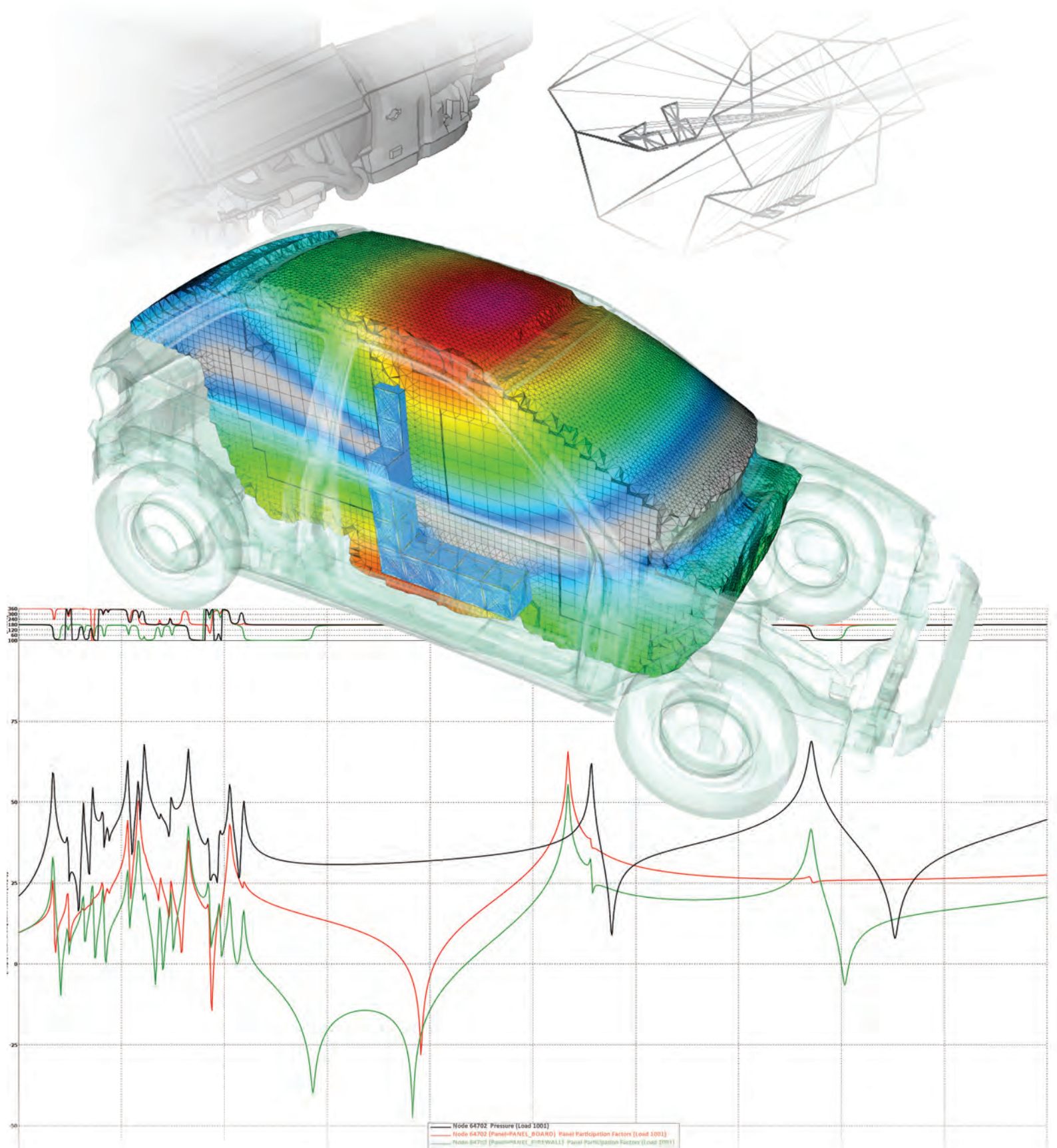
Benefits

- Cost and time-to-market reduction
- Novel features lead faster to results, while ensuring efficiency and quality

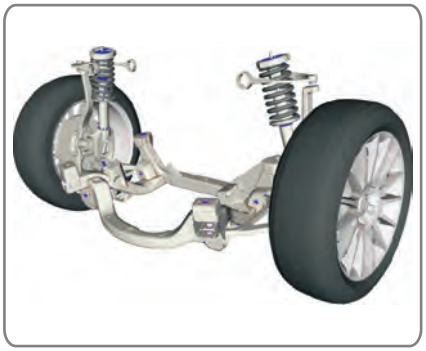


ANSA
µETA
PostProcessor
pioneering
software systems

advanced solutions for NVH pre- & post-processing

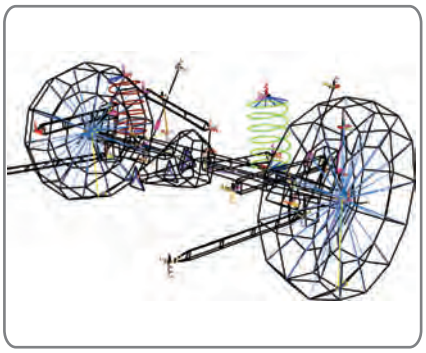


ANSA is a standard and robust solution for NVH simulation pre-processing that addresses the industry's needs for increased process efficiency and simulation results reliability.
The ANSA interoperable pre-processing decks, integrated with the advanced main features such as those of meshing, batch meshing, morphing, and coupling with optimizers, comprise a complete pre-processing environment that accommodates a wide range of functionalities. Also the Task Manager and the Data Manager, grant unique efficiency and repeatability during the build-up, as well as the updating and exporting of multiple high-quality models. Moreover, ANSA offers unique efficiency and repeatability during the build-up, update and export of multiple high-quality models. Furthermore, dedicated features serve the needs of NVH modeling in a unique manner.
ANSA efficiently supports popular codes such as ABAQUS/Standard, NASTRAN, ANSYS and PERMAS.



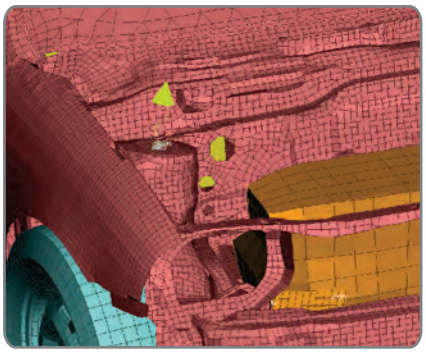
Model assembly and updates

- Visualization of model components and their changes in a varying manner through numerous navigation methods, such as per parts, per include file or per keyword
- Automatic generation and control over the components FE-representation, such as the conversion of parts or sub-assemblies to their reduced lumped mass representation, or to their replacement with the equivalent Modal Model
- ANSA also offers the comparison tool, that can be applied on parts or sub-assemblies level. Its purpose is to enable the decision making and application of model updates while retaining the model's integrity intact in cases, such as when re-applying the connections or the boundary conditions, and when re-distributing the masses.



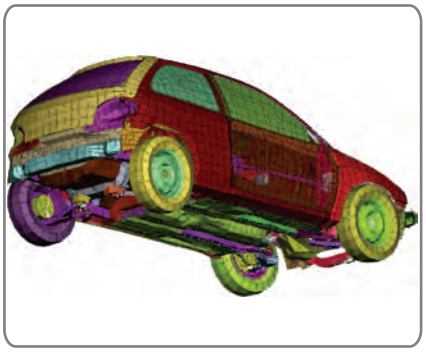
Connectors concept

The introduction of the generic concept of Connectors, carrying the detailed properties of the connectivity, allows for the composition of sub-assemblies. For demanding tasks, such as the replacement of one suspension with another, the Connectors are the definitive assistance for the new component assembly. The FE model representation is realized through library files that contain parameterized templates, applicable to any similar connectivity task.



Load case definition

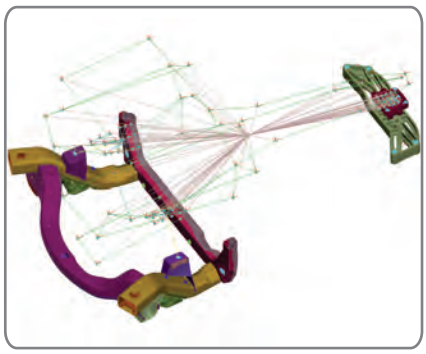
- Wizard-like assistants that accomplish complicated tasks, requiring minimum effort. Such features facilitate the definition of complicated keywords, boundary conditions and output requests. Their realization pattern can be stored and re-used for the build-up of the next model
- Special visualization technique for zero length line elements in an exploded mode and display of complicated beam cross sections, which enables the visual inspection of the model's components



Display model

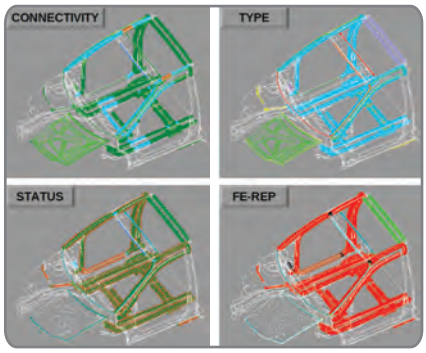
- Simplified model representation supporting the dynamic model reduction, for iterative Modal Response analyses. This can be achieved by creating display models (either with shell or PLOTEL elements) from a part, group of parts, sub-assemblies or even the whole model, with one shot
- A display model with shell elements serves, during post-processing with META, to visualize animations of the Eigenmodes which have been generated by means of modal superposition to the few DOFs of the nodes of the display model

Modal model creation



NVH CAE analyses can be very demanding in terms of time and CPU performance, especially for iterations with full vehicle models. Modal Models in ANSA can be used to replace a group of parts with its equivalent dynamic or its simplified model while retaining its full dynamic behavior. Such Modal Models are essential for FRF analyses and iterative runs, such as those for optimization purposes. Thus, in order to create a reduced dynamic model, an easy to use procedure exists for the automatic definition of a Modal Model, which creates its display model, launches the solver, and starts a META session for the calculation of the equivalent modal properties.

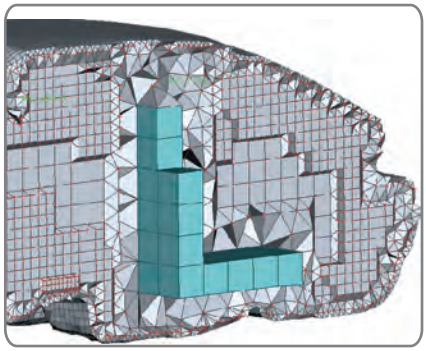
Weldings modeling



Welding models can be created through numerous semi- & fully automatic tools. Any information imported by PDM systems and any definitions created within the software constitute the base of the aforementioned models. ANSA offers a complete solution for:

- multiple connection types definition upon the model in one single step, through the Connection Manager, achieving:
  - Broad assortment of weld definition types for spot welds, adhesives, bolts or seam lines
  - Appropriate element definition with controllable quality
  - Depiction & enhancement of inappropriate connections information

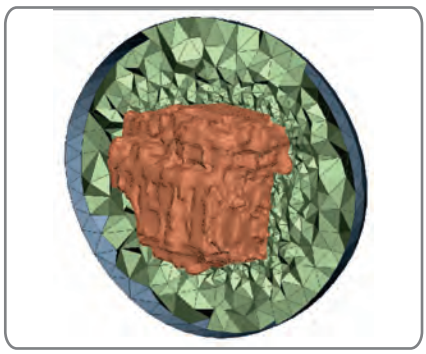
Cavity meshing for dynamic acoustic simulation



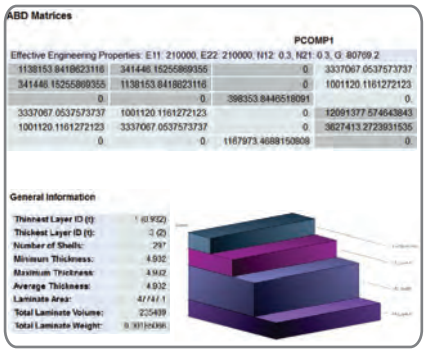
Along with all the integrated shell and volume meshing algorithms, ANSA provides meshing solutions focused on acoustics simulation. The new Cavity Mesher, especially designed for the fast creation of a volume mesh for an acoustic simulation, offers:

- Automatic detection and meshing of cavities, without the prerequisite of a closed volume
- An algorithm based on inner wrap methodology for accurate recognition of features from the structural wetted surface
- Automatic closing of holes and gaps with full and comfortable control of the identified openings
- A Hexa dominant or pure tetrahedral meshing algorithm
- Automatic creation of fluid properties
- Specification of sensor points as part of the created volume mesh
- Recognition of the seats and coupling of the Cavity mesh with them directly (paste) or indirectly (via MPCs)
- Optional creation of the NASTRAN ACMODL keyword and the respective fluid and structure sets to drive the fluid-structure coupling in NASTRAN
- Optionally generated PANELS for panel participation analysis in NASTRAN

Exterior Acoustics support



The preprocessing of the cavity for exterior Acoustic analyses has been improved significantly. Distributed script and semi-infinite area as template file supports the creation of adapted exterior cavity mesh for exterior Acoustic analyses for ACTRAN. Built in standard volumes, like spheres, boxes etc, enables the fast creation of the exterior cavity for any exterior Acoustic analysis solver. Especially with the newly supported Boolean functions the creation of adapted half-spheres is very comfortable.



Damped carpet modeling using laminates

A trend in modifying the damping properties of components is the usage of composite materials. These Damping Patches are meant to absorb oscillations, in order to improve the acoustic performance of the structure. This concept can be modeled in exploiting the integrated Laminate Tool, offering:

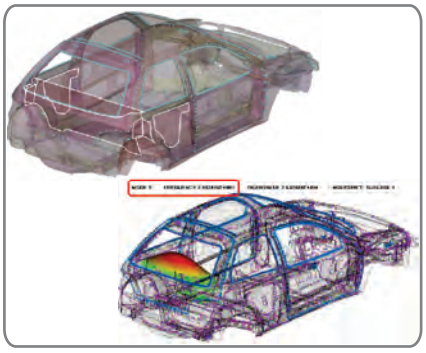
- High accuracy estimation of real composite characteristics of laminates
- Fast modification of each damping layer's properties
- Detailed reporting of the composite's structure and ABD stiffness matrix calculation

Mass trimming

The concept of Mass Trim items is introduced in ANSA, in order to address the problem of the proper representation of components by mass elements, while targeting on the correct mass distribution on the product. This can be achieved by:

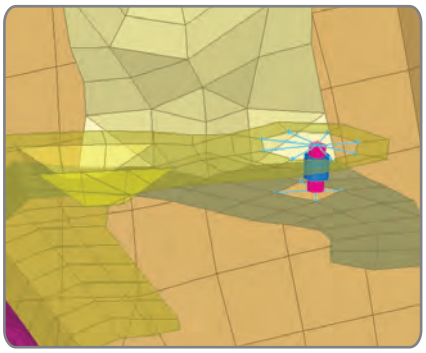
- Holding the information about the amount of added or target mass to be distributed
- Certain patterns available for distributing the mass area accurately, or property wise, according to each solver's requirements
- The representation or substitution on any FE component or part by its equivalent lumped mass with its mass and inertia properties automatically calculated
- Exploiting scripting, mass trimming information that can be imported and/or exported using ASCII, xml or other formats

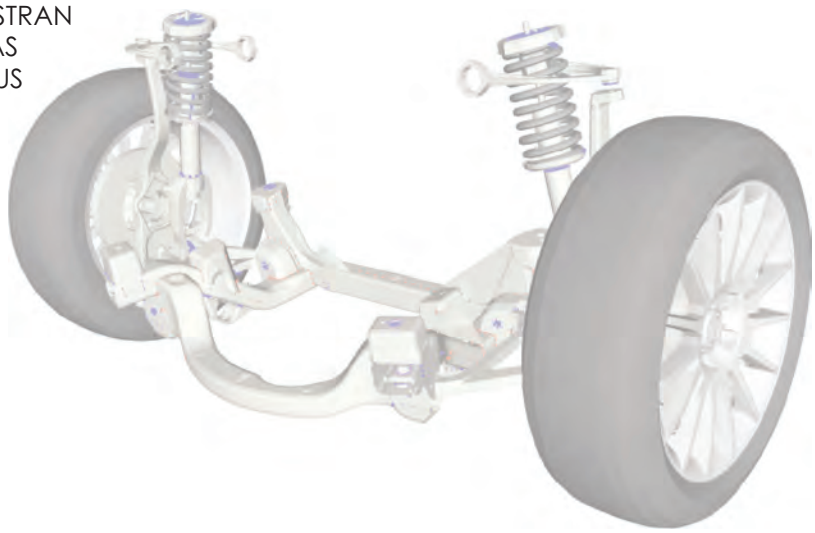
Analysis setup



The NVH model build up can be completed in one of many solver decks. Also comprehensive model checks, such as the appropriate definition of RBE3 DOFs to avoid singularities, the correct header definition and their references are available. Especially the new NASTRAN header definition functionality provides an effective tool to create fast and easy, even for low experience users, a complete header for all standard NASTRAN analyses. The generation of appropriate boundary conditions and solver specific elements is provided for several solvers such as:

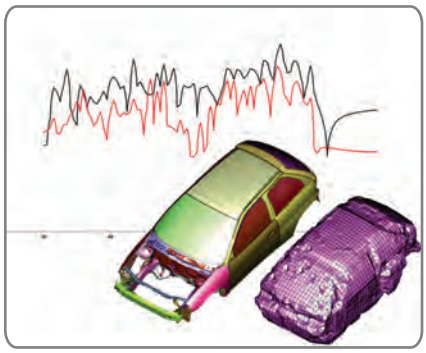
- MSC NASTRAN
- NX NASTRAN
- NE NASTRAN
- PERMAS
- ABAQUS
- ANSYS

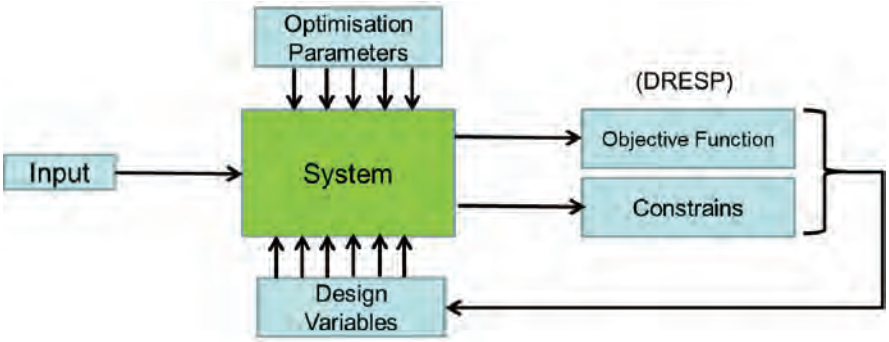


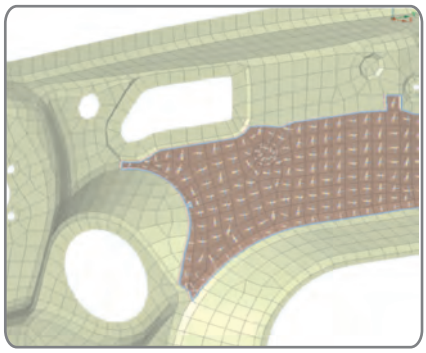


NVH Console

A comprehensive tool has been developed to streamline NVH analysis by conducting dynamic model reduction techniques of Include Components from full FE representation to Display Modal, ASCII Modal and FRF based model, from a single environment. The assembly of hybrid models of the above components, as well as the automatic creation of all connection entities at special defined hard points, is driven by a newly developed GUI as a net diagram view. Modal responses and operational animations beside of all standard NVH analyses, such as Transfer Path Analysis, are driven from within the NVH console by launching µETA-post.







Features

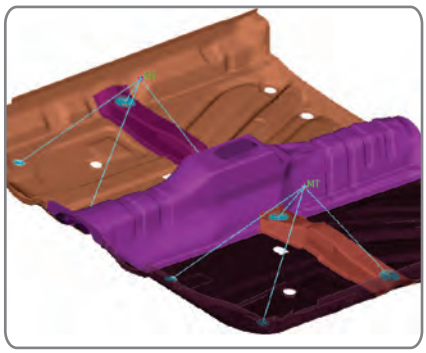
- Interoperable Decks
- Solver like entity cards GUI
- Process automation
- Model assembly
- Include files handling
- Connectors concept
- Modal Model creation
- Mass trimming
- Damped carpets modeling
- Acoustic cavity meshing
- Display model creation
- Quality validation & fixes

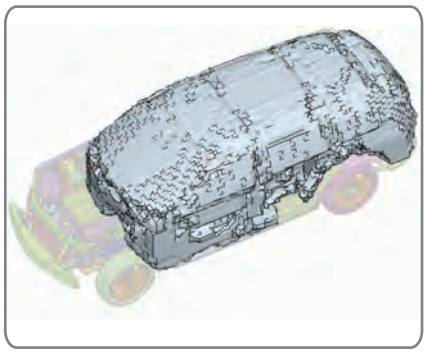
NVH console

- Drives pre-processing, solving and post-processing
- Dynamic model reduction of components
- Automatic connection of components
- Simplified overview of complex component models
- Error-free handling of large files
- Fast calculation of whole assemblies, suitable for "what-if-studies"

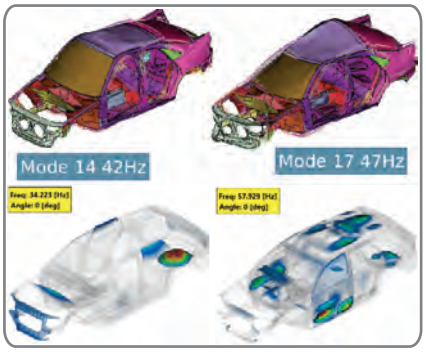
Benefits

- Supports the most important NVH tasks, including fast procedures for Modal Components building from pre-processing, solver initialization, to post-processing with µETA
- Easy to use cavity meshing and fluid-structure coupling within a single software environment
- High level of compatibility for common NVH tasks with ANSA & µETA
- Efficient modal model components handling
- Automation techniques that minimize costs and time to market
- Early-stage analysis iterations, and optimization possible with respect to NVH aspects and interface for most common optimizers



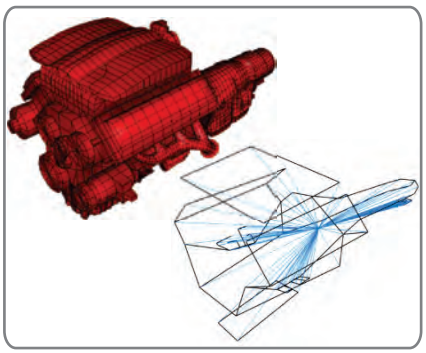


μETA brings a new dimension to the NVH post-processing by successfully addressing the bottlenecks and constraints involved. Its optimized performance for graphics memory usage and processing time along with a broad range of features and calculation options makes it an efficient environment for handling all NVH post-processing requirements.



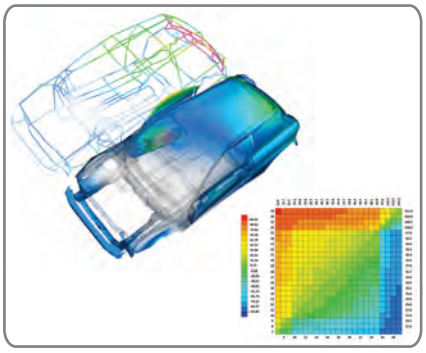
High-end software

NVH models constantly increase in size and complexity, hence challenging the hardware and software limits in many cases. μETA, with its unparalleled graphics performance and its proven effective model-handling, provides the means for smooth post-processing of big and complex models. Due to the optimized memory usage and the high data access performance, loading and processing a massive amount of data from large results files is not only feasible but also fast and productive.



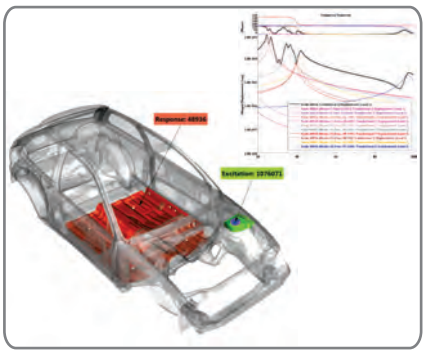
Modal modeling

In certain cases, analysts reduce the NVH models by substituting assemblies with their equivalent modal models. In this way, a considerable amount of solver-time is saved. These Modal Models can be built inside μETA out of the Eigenvectors results in just a few clicks through an intuitive interface.



Visualization of results

Contour display for 3D post-processing, as well as magnitude-phase, real-imaginary, polar 2D plots and contribution color-maps are used to support a broad range of modal response results including Panel participation factors, Normalized Grid Participation factors, Modal Participation factors, Acoustic Results and Mechanical & Sound intensity, with the latter calculated inside μETA. Different results for the same model can be easily displayed simultaneously in different windows, providing a complete overview of the analysis with the minimum memory requirements. The list of supported results is enriched with FEMZIP compressed Nastran results, Abaqus standard results, ANSYS results, test results in Universal format, as well as Design Optimization results.

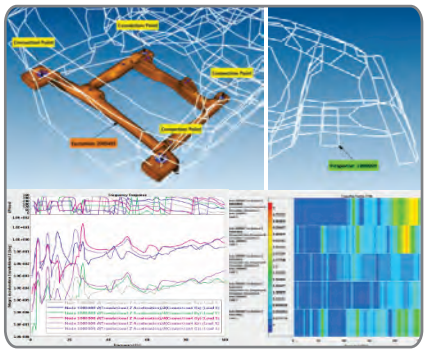


Calculation of modal responses & modal correlation

Frequency and transient structural and acoustic responses can be calculated swiftly and easily inside μETA, either as curves in the 2Dplot or as field data for contour display. The modal response calculator is fed with the Eigenvector results, while dynamic loads are either loaded from a file or defined inside μETA as well as the response DOFs. Existing structure-fluid coupling data can be used or simply generated quickly within μETA. Panels can be defined and panel participation factors can be calculated. The large amount of results created are handled easily through several grouping options. Modal correlation is performed through an easy-to-use tool and also MAC plots are created.

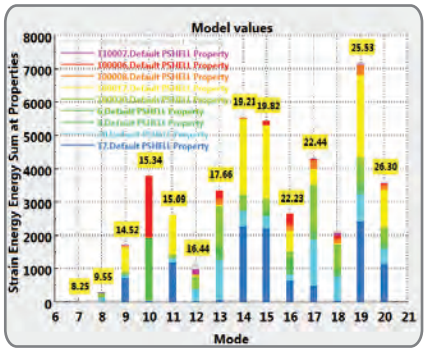
FRF assembly

The handling of hybrid models is available. Various components along with their results, either measured FRFs or calculated modes, can be connected inside μETA with either rigid or bushing connections. Modal responses are then calculated for the assembled model through simple interface. Connection forces can be plotted and transfer path analysis (TPA) can easily be conducted. Bushing properties are defined in μETA and their assessed sensitivities form the basis for deciding the design variables for the optimization procedure which is contacted using a generic optimization algorithm embeded within the tool.



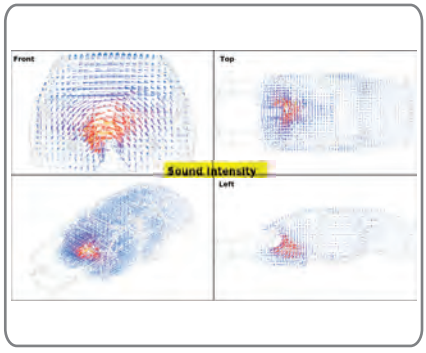
Other tools

A better insight of data can be obtained by using other general tools in μETA, such as the advanced filtering, the iso-contours, the cross-sections, annotations and the detailed statistics in spreadsheet form. A complete list of mathematical functions including Fourier Transforms and a broad range of filters facilitates demanding curve data processing. Powerful specific lists for connections, includes, boundary conditions, and sets allow for, among other, the effortless management of connections and the more efficient handling of the model. In the end, post-processing is finalized by saving the database in a μETA native binary format which can also be previewed with a free μETA viewer.



Reporting

The communication of results is further promoted through readily created reports either in html, PostScript or MS Office PowerPoint .pptx format. Reports can be filled in by dragging or dropping images, spreadsheets or other data from μETA to the report composer. Report templates can be used and pptx reports can be imported and viewed in slideshow.

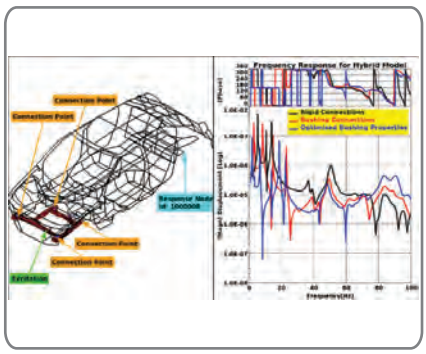


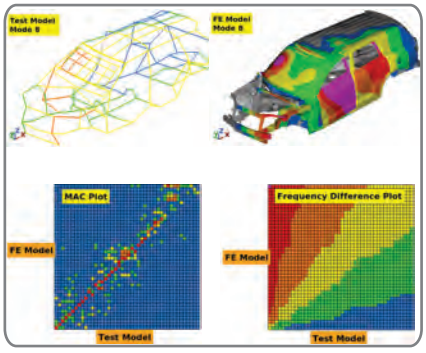
Features

- Outstanding performance
- Plot types: Magnitude-phase, real-imaginary, polar, contribution color-maps
- Mode correlation tool
- Modal Model builder
- Calculation of structural & acoustic modal responses, mode participation & panel participation
- Calculation of operating modes
- Calculation of mechanical & sound intensity results
- FRF assembly with TPA calculations capabilities along with embedded optimizer for connectors' properties
- Ultimate automation capabilities
- Complete reporting tool

Benefits

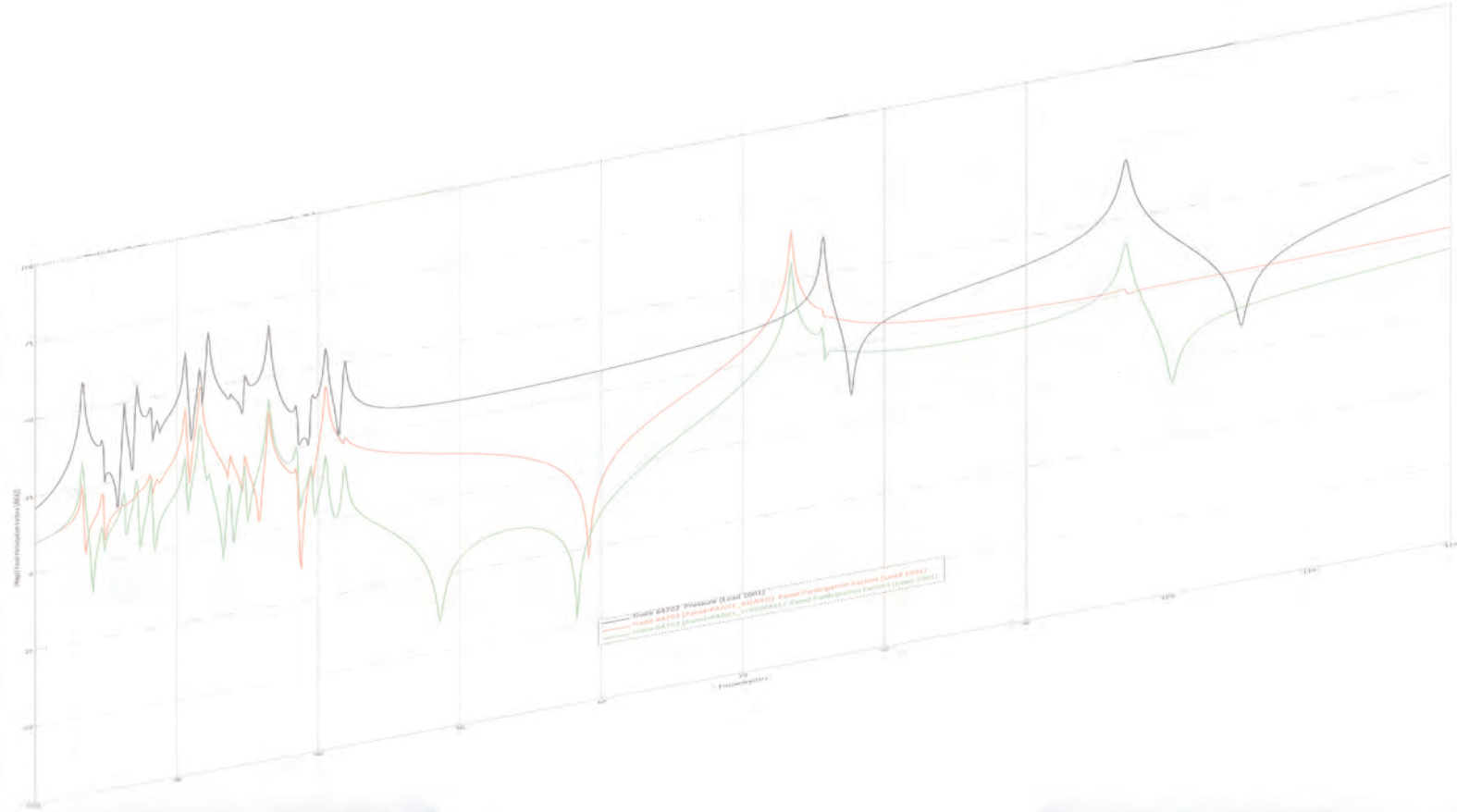
- Easy handling of large models
- Facilitates demanding results visualisation
- Results are reached within the minimum time through advanced algorithms for calculations
- What-if studies can be performed in no time
- Complicated repetitive processes can be automated and errors are avoided
- Significant time is saved due to high performance, automation capabilities, direct report editing, and standard toolbars

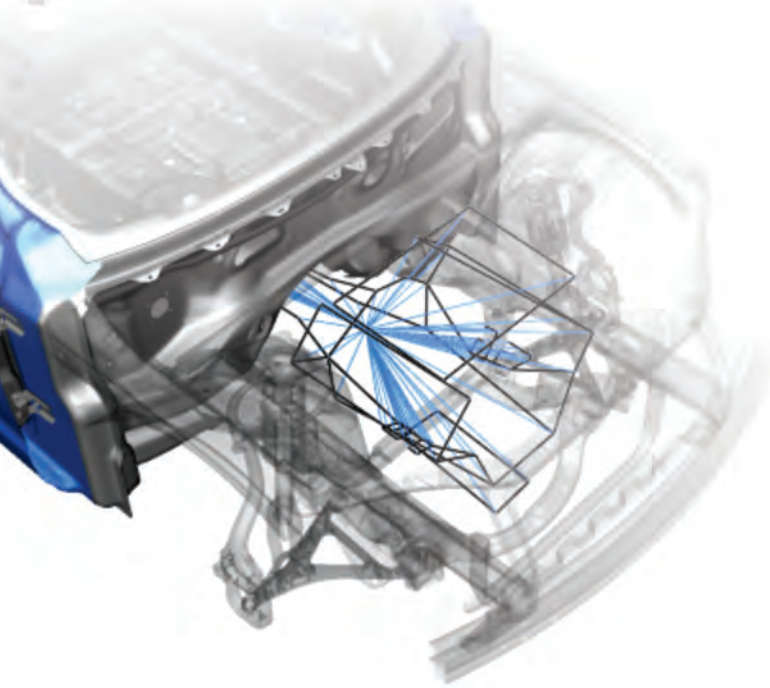


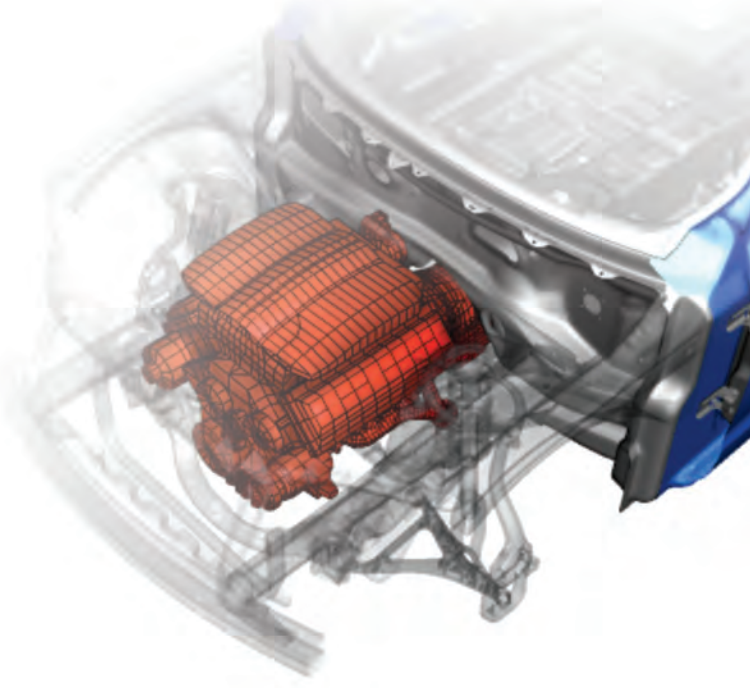


Automation

Last, a high level of process automation from results input to report output is achieved through parameterized session files, custom user toolbars and scripts, all of which can be easily created through intuitive editors embedded in μETA. Also, standard toolbars are available for facilitating a normal modes analysis through automatically creating a report, easily producing strain energy bar charts, and distinguishing global from local models. Furthermore, there are standard toolbars for reporting modal correlation results (MAC) and for Equivalent Radiation Power (ERP) calculation and post-processing.
The automation capabilities contribute to the easy and unhindered coupling of μETA to external optimizers. A simple-to-use toolbar, that streamlines the coupling, is provided with the standard package.

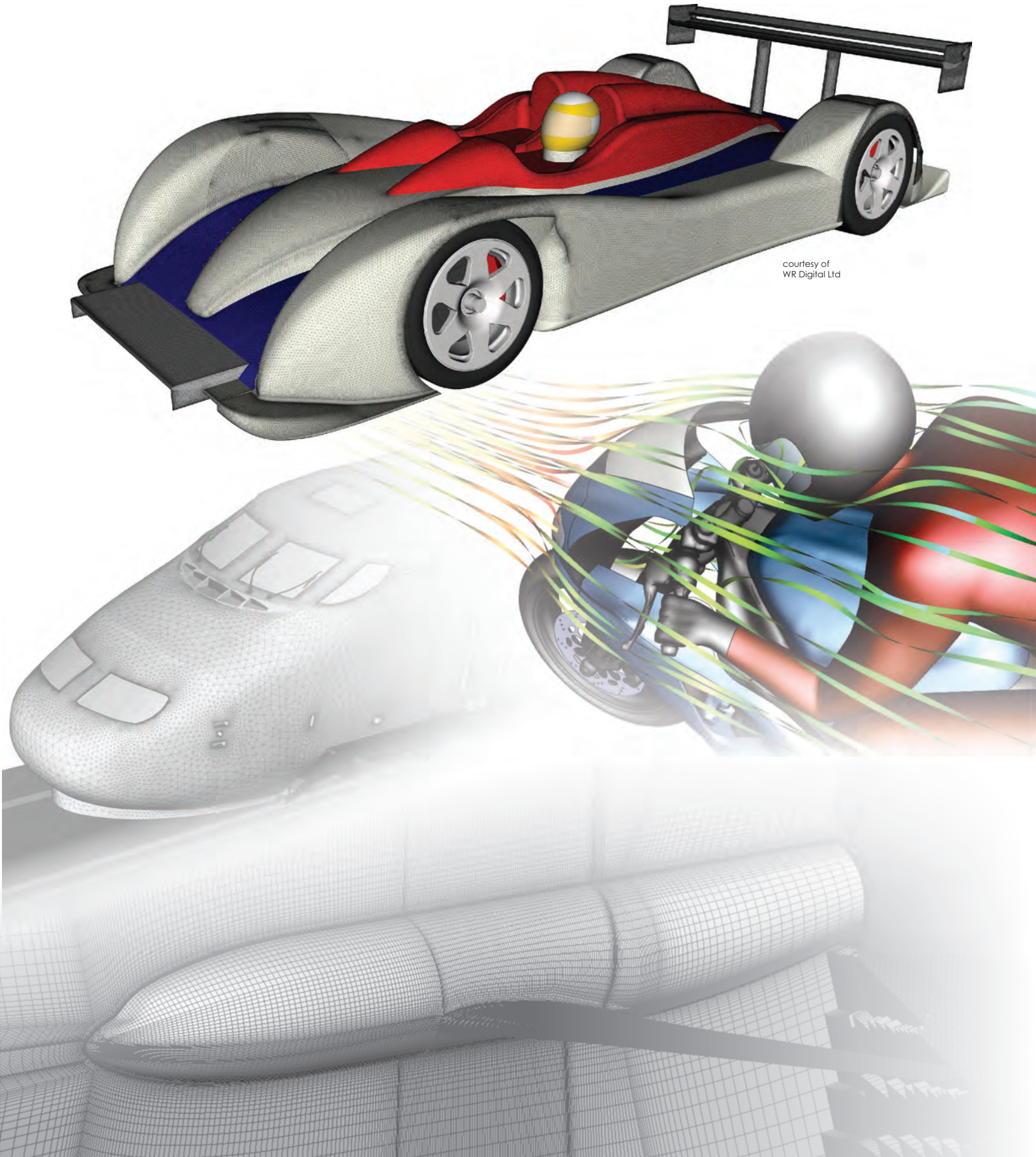




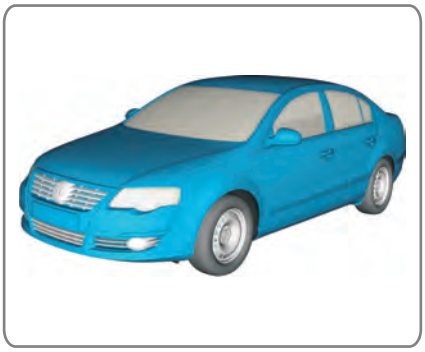


ANSA
µETA
PostProcessor
pioneering
software systems

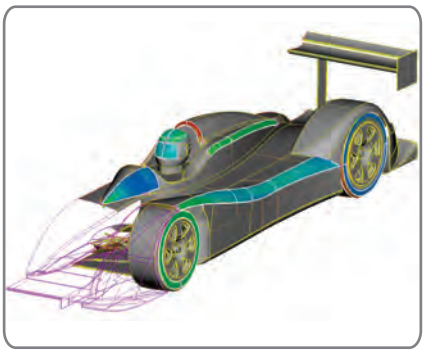
for demanding CFD pre- & post-processing


courtesy of
WR Digital Ltd

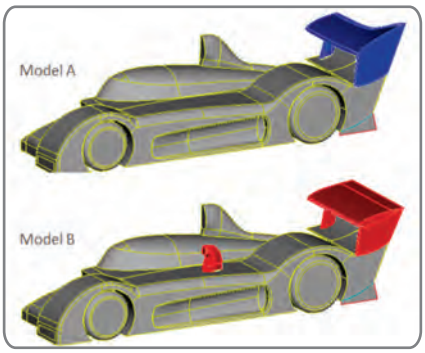
ANSA with its powerful functionality provides high efficiency solutions for CFD applications. Its capabilities meet the industry's demanding needs for external and internal flow simulations, increase productivity, and contribute to the high quality of CFD results. It is the choice of the leaders in CFD simulations in various sectors such as automotive, motorsports and aerospace among others.



Courtesy of Volkswagen AG



Courtesy of WR Digital Ltd



General features

- 32 or 64 bit code, for unlimited memory usage
- Multi-core CPU usage, taking advantage of all the hardware's CPU power
- Double precision for maximum accuracy
- Customizable interface with pre-defined CFD oriented layout
- CAD interfacing with neutral and native formats such as: IGES, STEP, VDA-FS, Catia v4 and v5, NX, Parasolid, PTC Creo Parametric, SolidWorks, Inventor, JT
- CFD mesh input/output for: ANSYS FLUENT, StarCD & CCM+, OpenFOAM, CFD++, CFX, SC/TETRA, UH-3D, CGNS
- Pre-processing and interfacing with all major CAE codes (NASTRAN, Abaqus, ANSYS, THESEUS-FE, RadTherm and more) and numerous neutral mesh formats (PATRAN, STL, VRML etc.)
- ANSA Scripting, with Python support, allows the automation of tedious pre-processing tasks, such as CAD data input, model structure build up, surface meshing and output, for increased productivity. It also supports the creation of user defined functions, further extending the software's functionality

Topology and CAD functionality

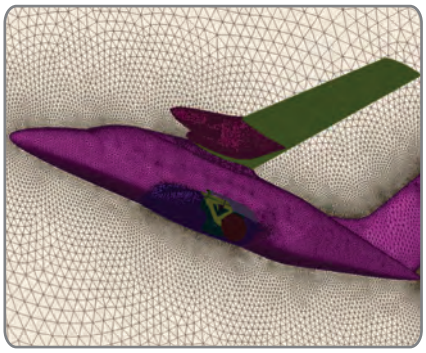
- Integrated CAD tools for geometry creation, modification, cleanup, defeaturing and watertight preparation
- Easy identification and isolation of inner or outer wetted surfaces, internal passages, zero-thickness walls, intersections, proximities and more
- Leak detection tools
- Automatic identification of similar geometry and substitution with virtual linked geometry. This, speeds up model build-up due to the interactive relation between the linked geometries

Model management

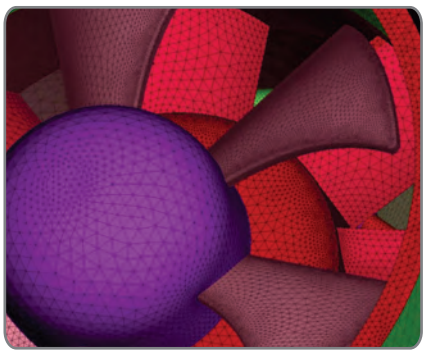
- Powerful model management through hierarchical part assemblies and properties, extracted from CAD input data and also modifiable in the ANSA Part Manager and Property list
- Novel comparison tool to simultaneously load two models and automatically identify their differences with respect to geometrical discrepancies or model characteristics like property names, batch mesh settings etc. Option to automatically replace only differences, allows for the quick update of the current model

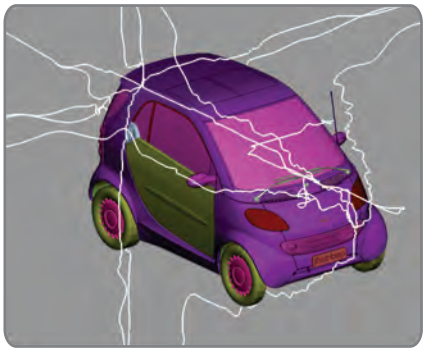
Surface meshing

- Automatic and robust mesh area simplification and defeaturing, by merging small surfaces into larger groups, enables optimum mesh quality according to user specifications
- Generation of tria, quad or mixed mesh using several meshing algorithms oriented to specific applications
- Fast CFD meshing algorithm for high quality surface meshing, with resolution adapted to local surface curvature, sharp edge features and user specified sources
- Generation of boundary layers for 2D analysis, and anisotropic meshing at high curvature at surfaces
- Flexible hexahedral or cylindrical Size Boxes for the automatic refinement of specific regions of the model
- Automatic and manual functions for shell mesh quality improvement
- Powerful reconstruction algorithm used for the improvement or modification of surface mesh, subjected to user-specified requirements (length, element type etc.). Applicable either during geometry based mesh generation or to imported shell mesh
- Shell mesh clean-up tools (close openings, paste nodes, connect elements, intersect meshes etc)
- Powerful and versatile tools for handling and combining CAD geometry and imported shell mesh
- Quality check according to numerous criteria for various solvers
- Clear identification of poor-quality elements, colored by criteria type or value
- Contour plot of mesh colored according to mesh distortion or mesh quality
- Mesh integrity checks (unmeshed areas, intersections, free edges, proximities etc.)
- Detailed mesh information and quality statistics.



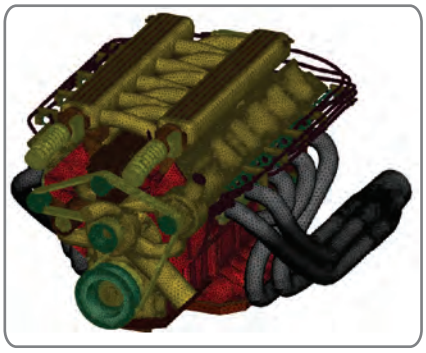
Courtesy of Evector, spol. s r.o.

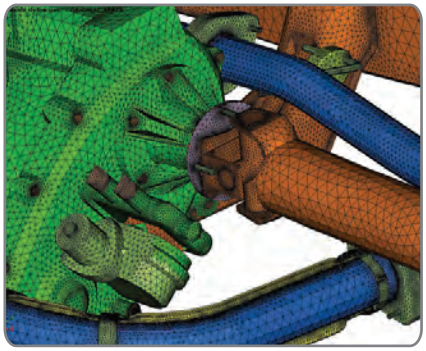


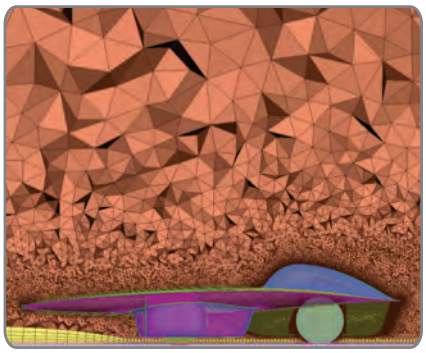


Wrapping

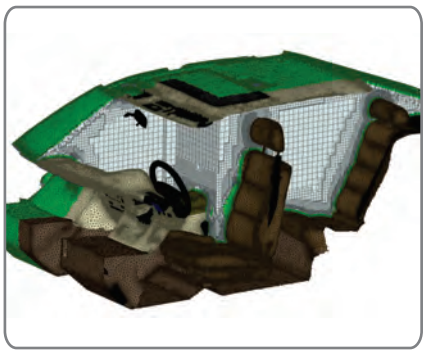
- Powerful surface wrapping tool, allows for the creation of a fully watertight model at a fraction of the time that would be required in the traditional surface meshing approach, regardless of the complexity of the geometry
- Specification of outer or inner wrapping. Domain selection by largest size or through seed point specification.
- Advanced wrapping algorithm that captures all feature lines of the model, with curvature and proximity refinement, variable length, and per property user defined parameters. Size refinement boxes are also applicable
- Intelligent leak detection tools with multiple seed point specification and automatic identification of all leak areas. Advanced manual and automatic tools for leak closure

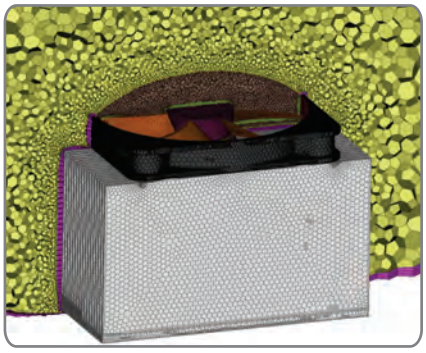


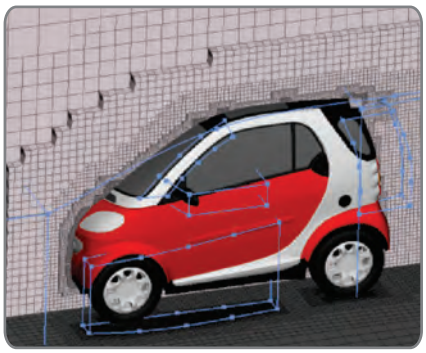


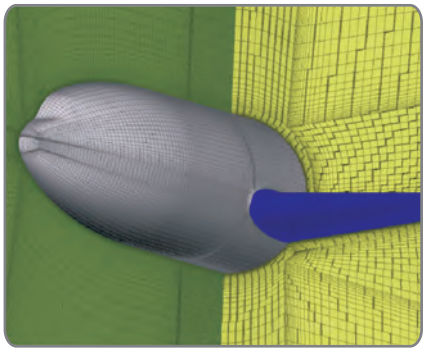


Courtesy of Actiflow BV



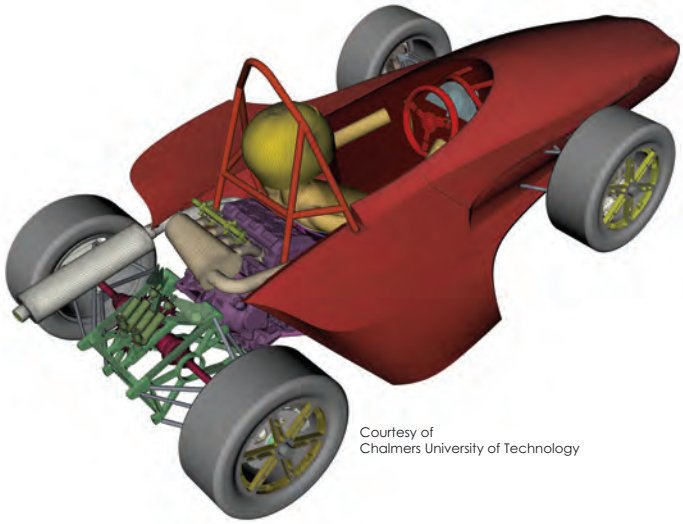






Volume meshing

- Generation of penta and hexa boundary layers with variable parameters per property, advanced controls for squeezing, collapsing or excluding, to overcome quality and proximity problems, generation of layers from both sides of zero-thickness walls and more. A very robust smoothing algorithm ensures high quality layers generation all around complex model geometries
- Automatic Volume detection and definition algorithm. Fast and robust Volume meshing for tetra, prism, pyramid and hexa elements
- Conformal, variable size, Hexa-Interior/Hexa-Poly mesh, aligned to local coordinate systems
- Flexible hexahedral or cylindrical Size Boxes for tetra and Hexa-Interior/Hexa-Poly with controlled mesh refinement and growth rate in space
- Unstructured hexa and penta meshing through map and sweep algorithms
- Pure hexa meshing based on multi-block decomposition of geometry with associated box topologies
- Polyhedral mesh generation though conversion of hybrid mesh
- Octree trim-hexa/polyhedral meshing algorithm applicable to non-watertight models for quick generation of volume mesh



Courtesy of
Chalmers University of Technology

Features

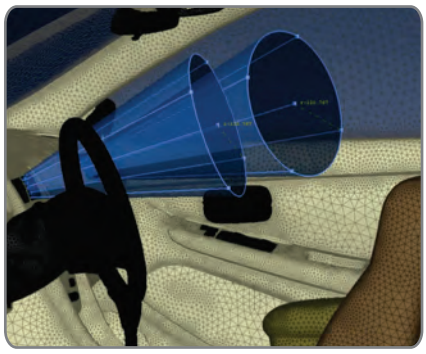
- Geometry clean up and de-featuring
- Watertight preparation
- Shell & Volume meshing
- Surface wrapping
- Boundary layer meshing
- Batch meshing
- Hexablock meshing
- Model checks & fixes
- Mesh & geometry morphing
- Numerous CAD/CAE interfaces
- Coupling with optimizers
- Python scripting language

Benefits

- Covers all the CFD pre-processing needs in a single environment
- Provides and combines both high quality and "push button" meshing approaches
- Eliminates the use of task-specific software
- Minimizes cost and time to market
- Novel features lead to results faster, ensuring quality and consistency
- Common preprocessing platform for numerous CFD & FEA codes

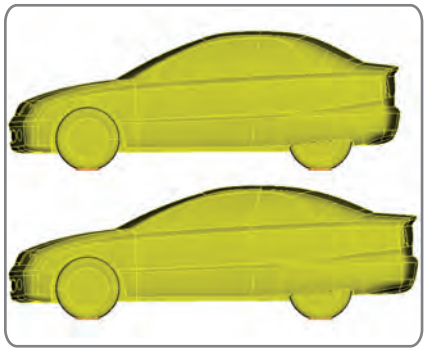
Batch meshing

- Complete automation of all the steps of CFD mesh generation based on pre-defined scenarios, for surface meshing, wrapping, layers generation and volume meshing
- A process that can be applied repeatedly on new geometries, based on part or property name filtering conventions, ensuring mesh consistency and saving time and resources



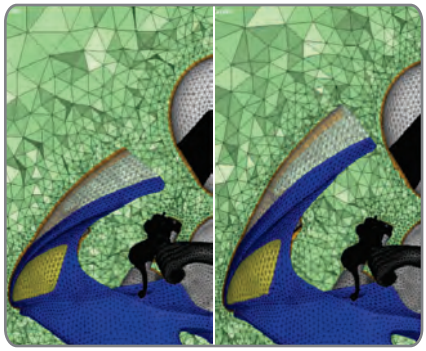
Model setup

- Model validity checks with customizable templates of several checks available per specific task, through the Checks Manager
- Specification of boundary condition types for ANSYS FLUENT, Star CD/CCM+ and UH-3D
- Complete solution setup for OpenFOAM cases, including initial and boundary condition specification, physical and numerical parameter set-up, and solution controls
- Support also for THESEUS-FE and Radtherm model files setup



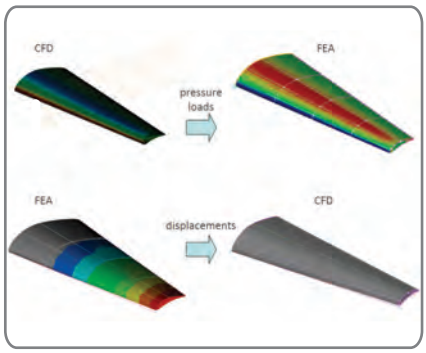
Morphing & optimization

- Flexible parametrization of your model
- Fast and controllable Morphing of surface & volume mesh of large and complex models through Morphing Box techniques
- Direct Fit Morphing (DFM) for quick shape optimization without morphing boxes applicable to mesh and CAD geometry
- Integrated tool in the same environment with all the other pre-processing functionalities of ANSA
- Fully automated batch process coupled with various optimization software and CFD solver
- Support of Adjoint solver sensitivities based optimization



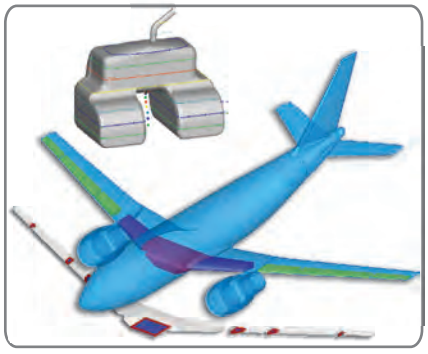
CFD FEA coupling

- Ability to map pressure loads from CFD analyses to different FEA meshes, through the ANSA Results Mapping tool
- Ability to map FEA calculated deformations back to CFD meshes, through the Deformation Mapping tool

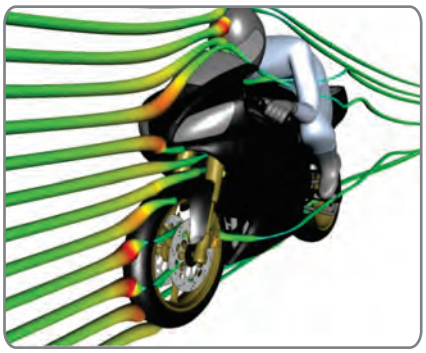


Liquid level calculations

- Fast calculations of liquid volume, levels and CoG positions for liquid tank systems
- Detection of resting and unused liquid areas
- Also applicable for initialization of two-phase flow simulations

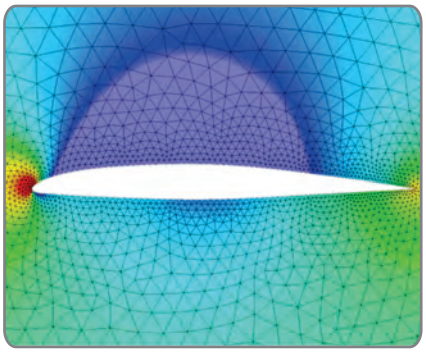


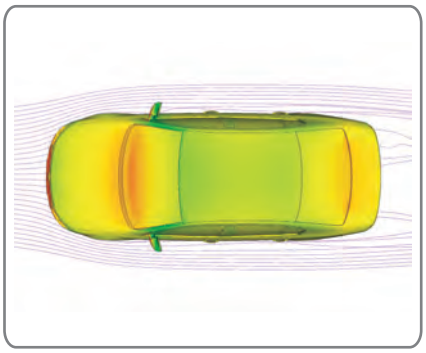
μETA, the leading post-processor in structural analysis, extends its support to CFD codes. μETA's indisputable high performance capabilities enable engineers to easily handle and explore extremely large and complex models. Through the numerous validated analysis tools and automation capabilities, engineers avoid time-consuming data mining and focus their engineering judgement on important facts.



CFD results visualization, correlation and reporting, benefits from the high-performance and multitude of μETA's features and tools that are already successfully deployed in the structural analysis field.

Some of the main features of the software are:
- Extremely fast reading and handling of large data sets with low memory footprint
- Complete automation and customization from results input to report creation, through session files and scripting
- Powerful graphics for the display of contour plots, iso-surfaces, cut planes, streamlines and vector plots
- The easy model handling through Properties and Groups like in ANSA
- Full domain representation for symmetrical and periodic simulations
- Surface intergrals and forces calculations
- Identification of point data in arbitrary position on or off the model
- The ability to save selected results in native μETA format, reducing the amount of stored data
- The query of model dimensions
- The query of highest and lowest values of flow variables, and their location
- The superimposition of annotations, enables the quick extraction and display the exact information needed
- A notable strength of μETA is its capability to load and process more than one simulation model simultaneously for correlation studies. Differences on the results between different CFD solvers, geometries, meshes or numerical setups can be easily identified
- High quality images and animations can be created and inserted in .pptx, .pdf and .html reports, through the report composer tool



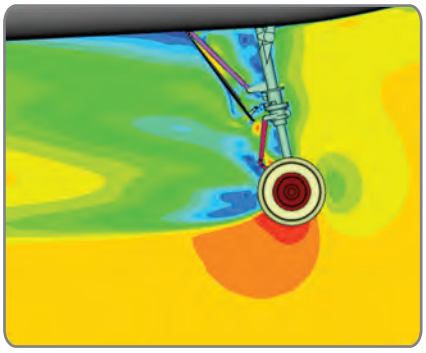


Supported formats

- ANSYS FLUENT
- OpenFOAM
- STAR-CCM+
- CFD++
- SC/Tetra
- Ensight

Contour plots

- Display of results as contour plots on model surfaces
- Contour plots cutting planes and iso-surfaces

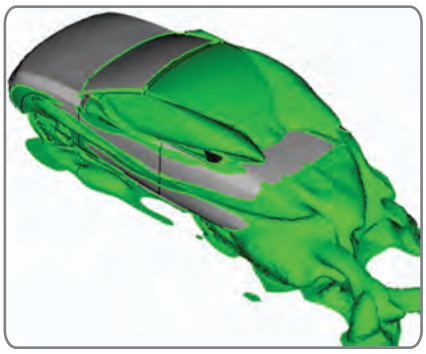


Vectors

- Display of vectors and vector components on any surface

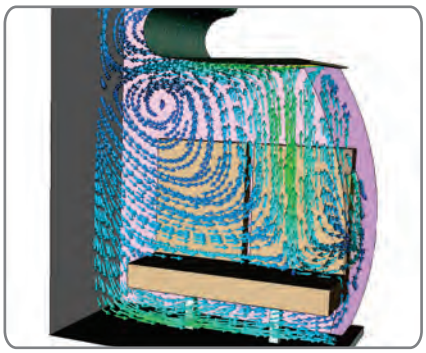
Streamlines

- Draw streamlines as lines, ribbons, cylinders and also animated particles and arrows
- Colour, twist and modulate streamlines by any available variable
- Oil flow visualization



Display

- Interactive view control
- Results animation
- Orthographic or perspective views
- Stereoscopic viewing (using special equipment)

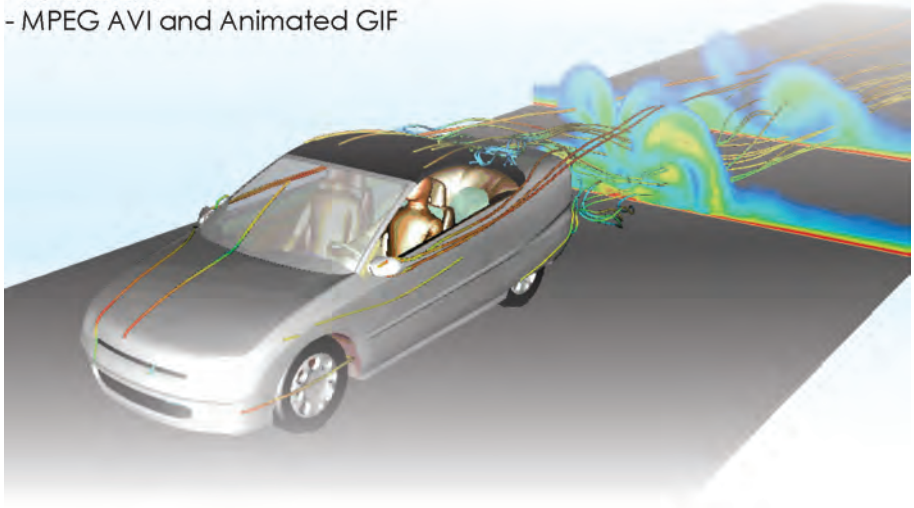


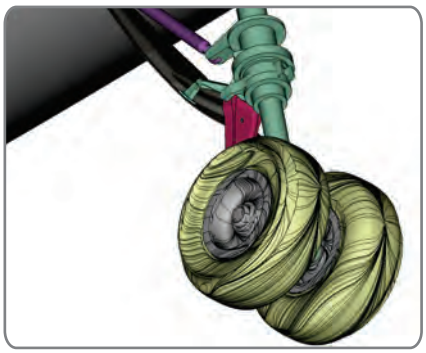
Automation

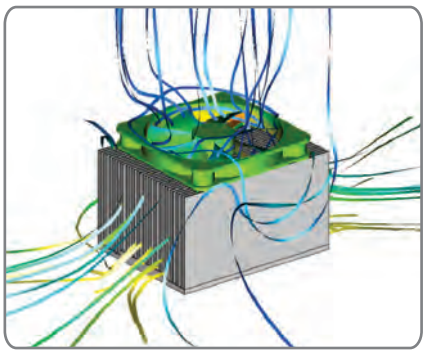
- Session files
- Python scripting language support
- User toolbars
- User variables

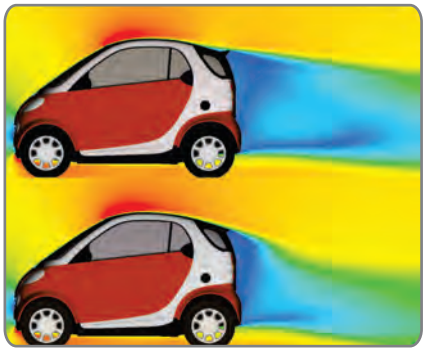
Image and video output

- Popular image formats : JPEG, PNG, TIFF, BMP, GIF
- PS and EPS
- MPEG AVI and Animated GIF







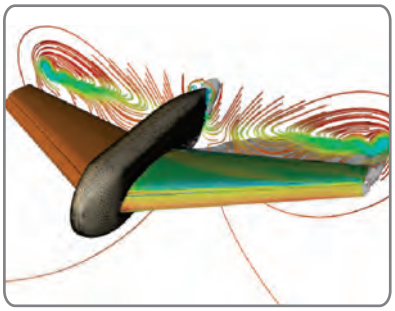


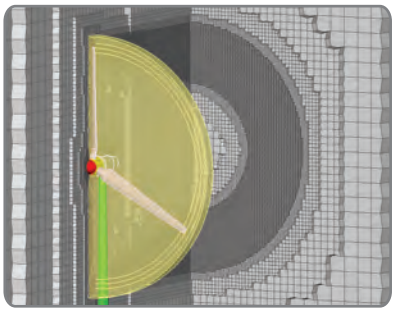
Features

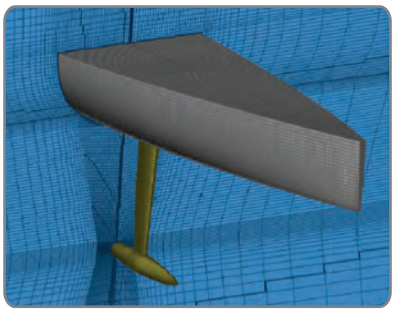
- 3D & 2D post-processing
- Iso countours, Cut planes, Vectors, Streamlines
- Multiple model handling and comparison
- Numerous interfaces
- Process automation
- Python scripting language
- Parameterized sessions
- Video & image correlation
- Annotations
- Reporting
- Native database
- Free viewer

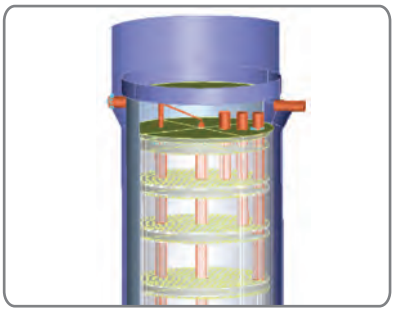
Benefits

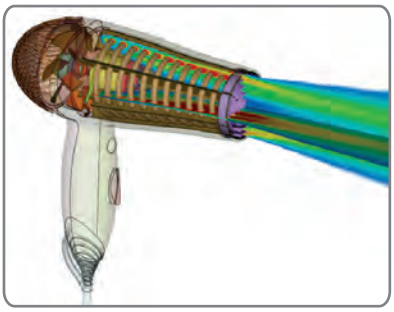
- Extremely fast reading and processing of large data sets ensure maximum productivity
- Automation techniques that minimize cost and time to market
- Common post-processing platforms for numerous CFD and FEA codes

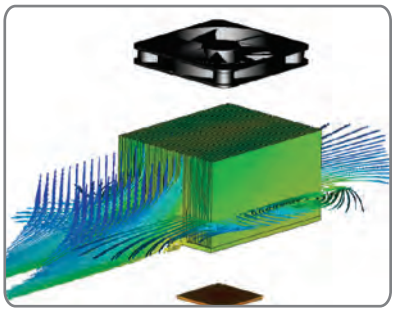


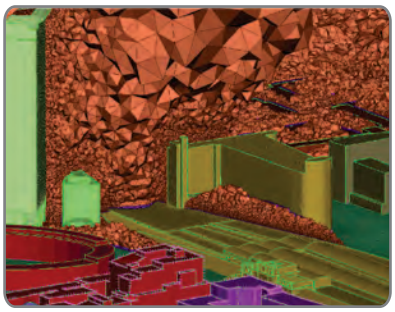


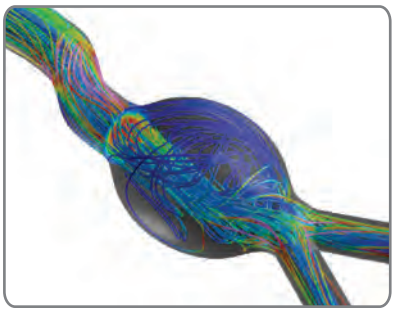






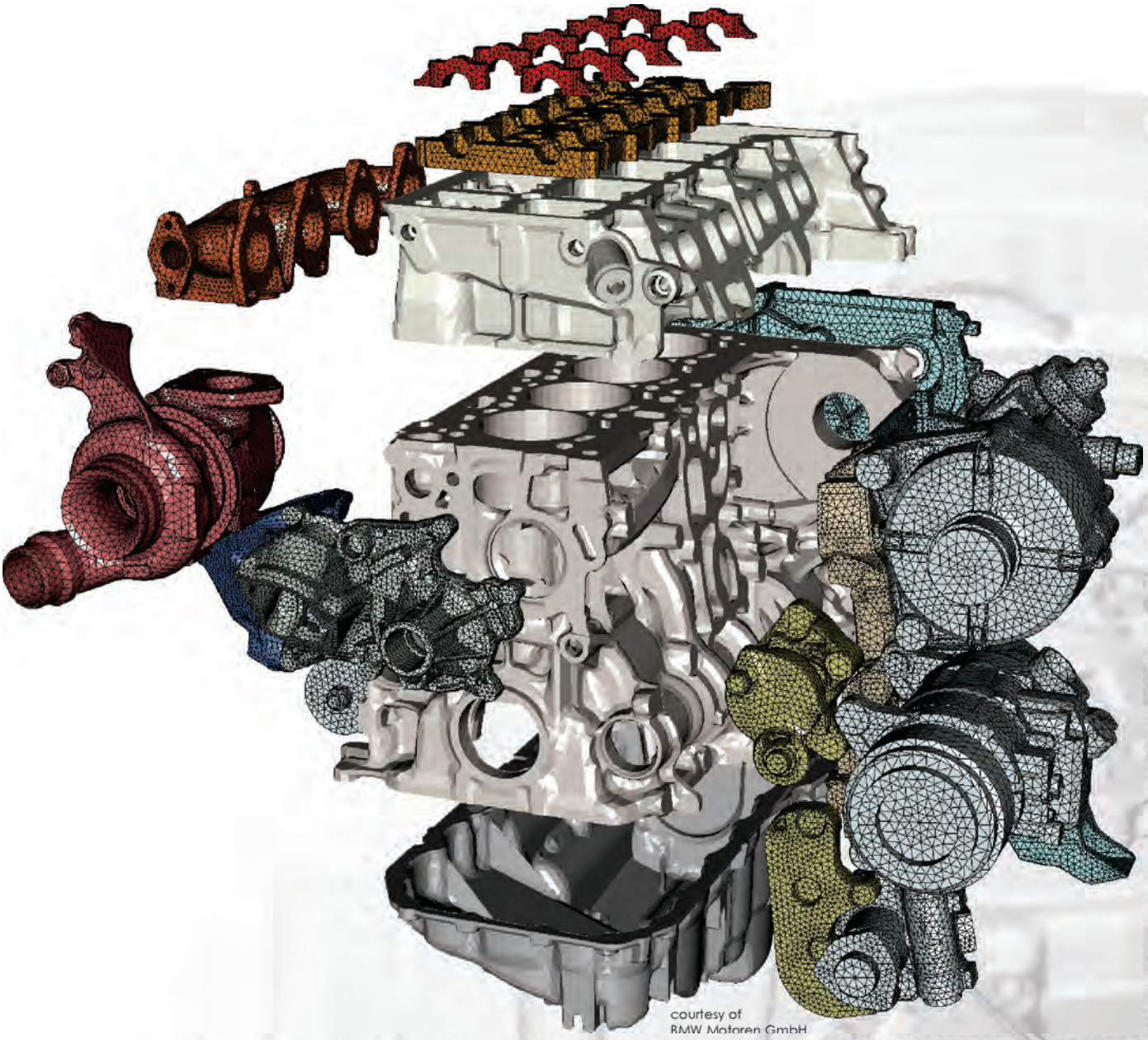






ANSA
µETA
PostProcessor
pioneering
software systems

optimum pre- & post-processing for powertrain

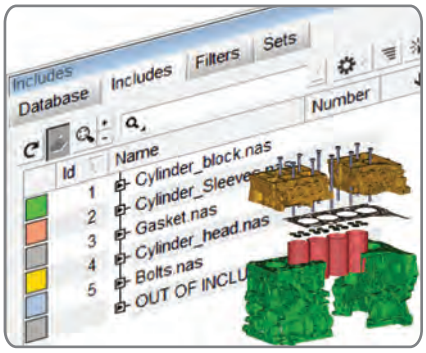


courtesy of
BMW Motoren GmbH

With its complete solutions for meshing, assembly, contacts definition and boundary conditions set-up, ANSA becomes the most efficient and effective solution for powertrain pre-processing.
An easy to set-up, fast and robust, meshing algorithm (Solid structural mesh) that ensures very accurate capturing of geometrical features, and a powerful and versatile assembly center for the generation and handling of bolt models have been specifically developed to satisfy the most demanding powertrain CAE teams.

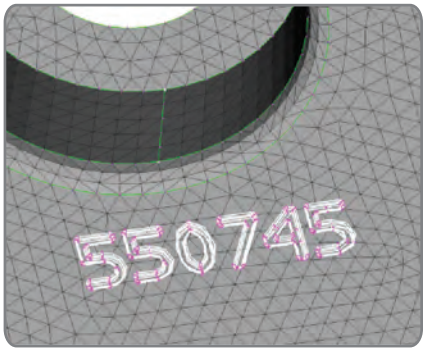
Translation

With ANSA, the information ported from CAD is not limited to the geometric descriptions of the components, but extends to the product structure together with several μETA data, such as the name, the number and version of each component, its position matrix, and its material information.



Set-up of meshing

The mesh result is dominated by 4 parameters:
a. Target element length
b. Distortion angle and distance (chordal deviation)
c. Maximum aspect ratio
d. Minimum element length
Feeding these parameters to the Batch Meshing Wizard automatically generates the appropriate mesh parameter and quality criteria files.

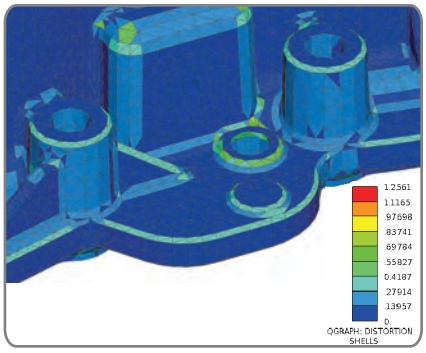


Feature recognition

Geometrical features are automatically detected ant the feature treatment is easily defined. Fillets are automatically identified by their shape and continuity, requiring no radius and width specifications. Tubes are also recognized likewise, with no need for additional input. Also, flange areas are detected based on parts proximity and shape.

Geometry simplification

Narrow geometrical faces are automatically removed, generating wider and better shaped meshing areas. Imprinted and protruded logos can be totally removed. In the case of narrow ribs, their sharp edges are automatically recognized and maintained.



Fillets treatment

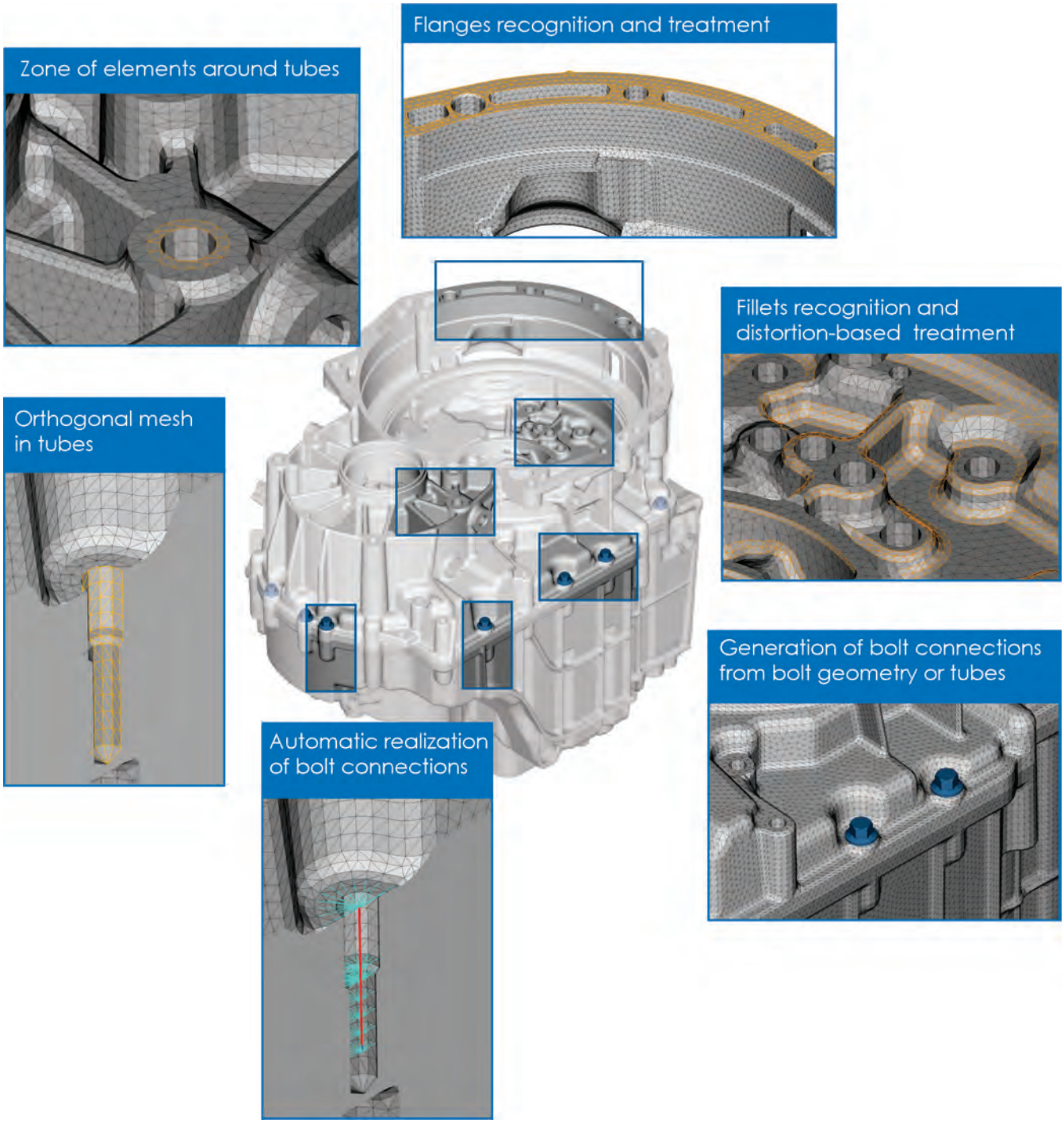
Number of elements in the fillets' width is controlled by the designated distortion and minimum element length, ensuring precise geometry representation with the minimum input requirements. Along the fillets, the element length is controlled by the prescribed quality criteria of aspect ratio and skewness, or by setting a local target lenght. The user can select among orthogonal and equilateral trias arrangement.

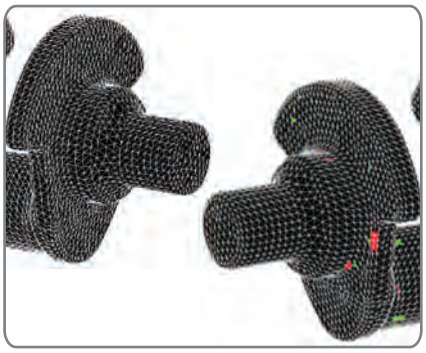
Tubes treatment

The number of elements along the tubes' perimeters is controlled by the designated distortion and minimum element length and is maintained along the tubes' height with the use of orthogonal trias. Zones of orthogonal or equilateral trias and user-specified width can be generated around the top and bottom rings of the tube. Narrow tubes can be filled.

Flanges treatment

The mesh density on solid flanges can be increased to assure a contact of better quality. At the same time the meshing algorithm ensures that no flange mode is moved away from the surface during the internal phase of quality improvement.



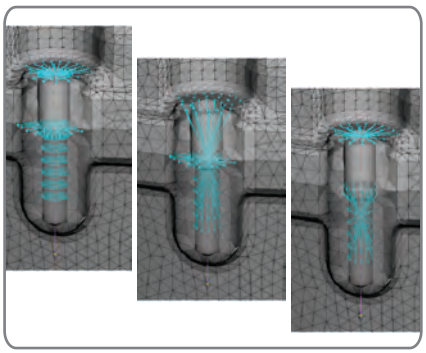


Second order mesh

The 2nd order nodes are always located on the geometry and not on a straight edge. This way, the component's shape is most accurately represented and the deviation of the CAE model mass from the actual mass is minimized. The elements' quality is efficiently controlled with the jacobian criterion.

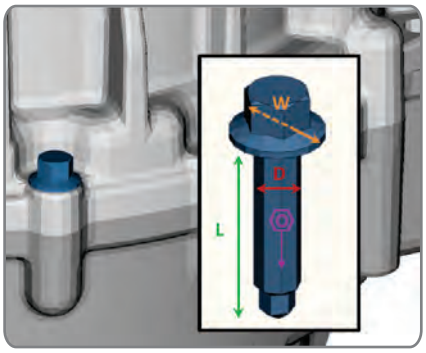
Mesh quality improvement

Several tools are available for the check and verification of the mesh quality, from the on-screen visualization of quality violations, to the automatic generation of quality reports. Distortion and distance from geometry graphs are offered to assure the precise capturing of geometrical features. For all those cases, when quality improvement is necessary, very powerful automatic tools are offered to handle both shells and solids, either by slightly moving nodes, or by local reconstruction of the mesh topology.



Bolts modeling

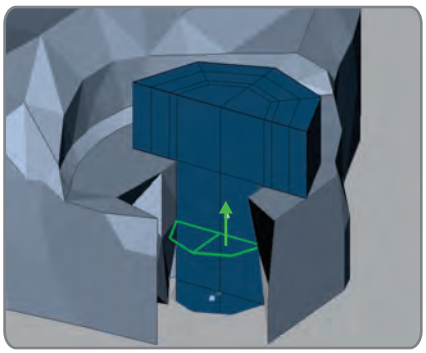
Reduced representations of bolts, consisting of any combination of elements, are generated with the aid of bolt connections. A bolt connection holds all the important information of bolt joints like the diameter, the head or washer diameter the length and the connected parts. Bolt connection entities can be transferred from one model to another through connection files and are realized into suitable FE-representations with the aid of the Connection Manager.



Bolt connections generation

Bolt connections can be automatically generated from bolt geometries, inheriting from the latter geometrical attributes like the location, orientation, diameter, head diameter and length. In case of lack of the bolt geometries, bolt connections can be automatically generated from bolt holes and tubes. In either case, the bolt connectivity is detected by performing a search in the bolt's vicinity, and is expressed either with part numbers or property ids.

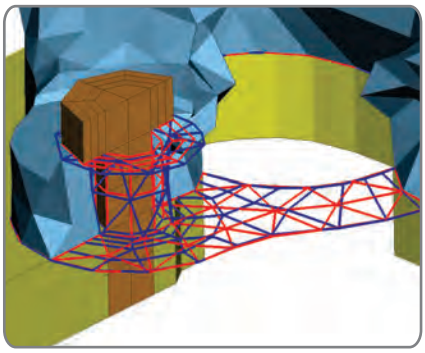
Bolt connections realization

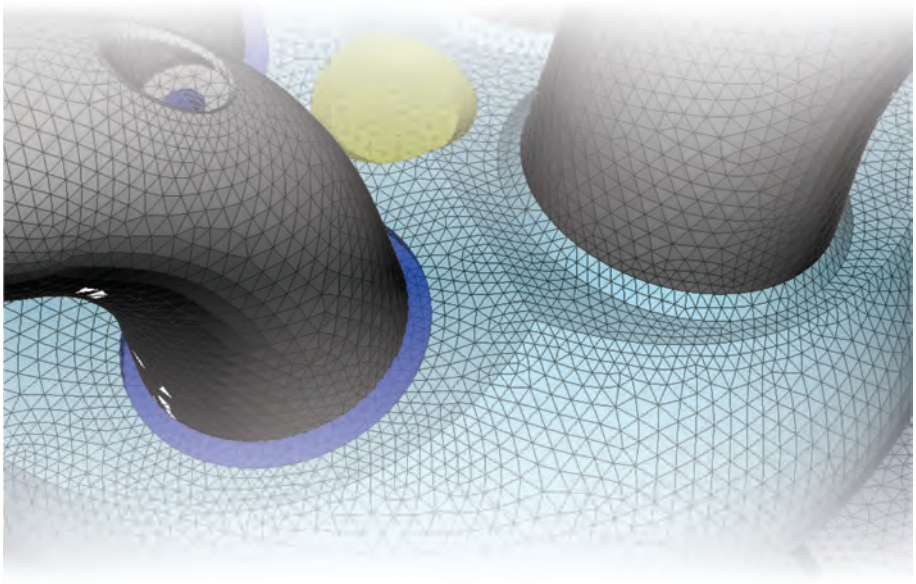


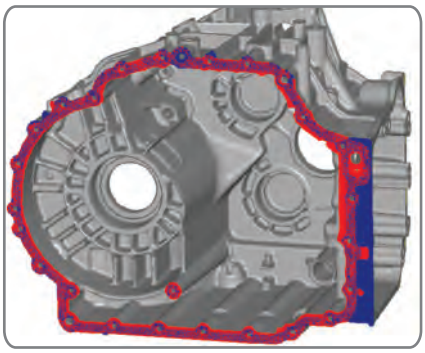
With the aid of the Connection Manager, multiple bolt connections can be realized into suitable FE-representations in a single step. A multitude of bolt FE-model options are available for the generation of the bolt body and its attachment to the connected parts. Bolt connections are automatically re-applied after part replacement, allowing the effortless update of the assembly.
Pre-loading of the bolts can be easily defined, either with the set-up of pre-tension keywords -for the solvers that support such- or with the definition of thermal loads, using the thermal expansion properties of the material.

Contacts definition

Contact areas between solid parts are automatically detected based on proximity. Master-slave contact constraints are directly defined either by using contact card templates or by the default values, and with the contact clearance automatically updated.





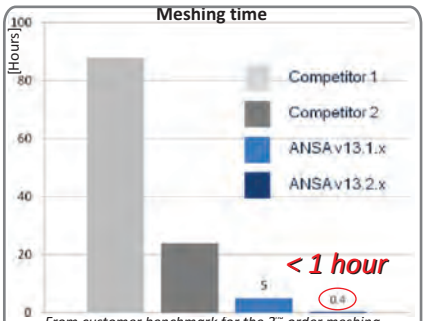


Features

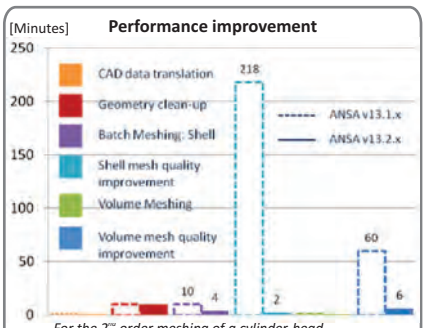
- Include-files handling
- Batch mesh dominated by target element length, distortion angle and distance, maximum aspect ratio and minimum element length
- Geometry simplification
- Fillets, Tubes, and Flanges treatment
- Second order mesh for more accurate representation
- Mesh quality improvement
- Bolts modeling through reduced representations consisting of any combination of elements
- Automated bolt connections generation from bolt geometries
- Multiple bolt connection realization into FE-representation in a single step
- Automatic contact definition
- ANSA can be easily coupled with any parametric optimizer
- FE & geometry morphing 2D & 3D parametric morphing

Benefits

- Time-efficient pre-processing that minimizes cost and time to market
- High end solutions for meshing, model assembly and boundary conditions set-up
- Configuration files and templates allow for the standardization of the pre-processing steps ensuring repeatability and enforcing robustness



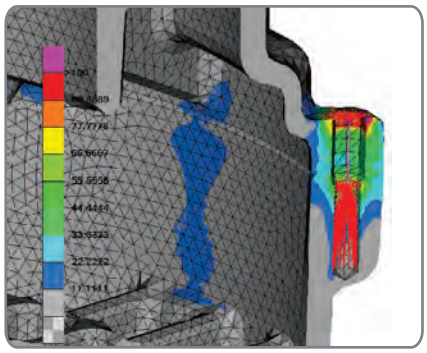
From customer benchmark for the 2nd order meshing of a cylinder-head



For the 2nd order meshing of a cylinder-head

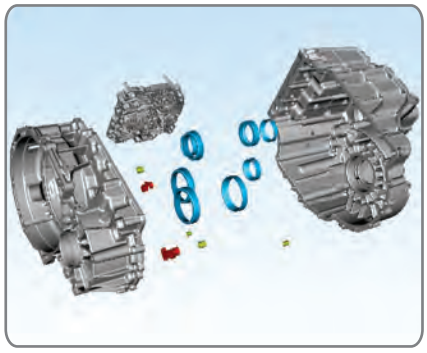
μETA is a complete solution for multi-disciplinary post-processing that stands out due to its model organization capabilities, the variety of tools for the assessment of the results, the available functionalities for the identification of the areas of interest and hot spots, the tools for direct calculations based on solver results, the options for the communication of the results and for its automation capabilities.

Model navigation

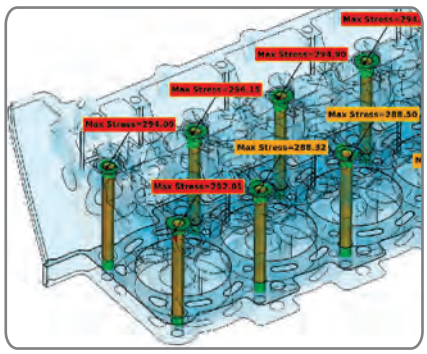


μETA offers model organization capabilities that are crucial in handling complex assemblies. These capabilities are made possible through the Parts Manager, Connection , Sets, and Properties and Material lists. Model structure connection and information are passed through ANSA comments in μETA. The sets and the loads from the result file are listed in the Sets list. Properties and Material lists are also available, in a similar to ANSA way.
A variety of tools are offered for the assessment of the results. Such tools offer:

- Exploded view that provides with great assistance in the visualization of the results when contact pairs are involved and results need to be viewed on the different contact surfaces
- Cutting planes which is especially useful when solid elements are involved
- Fringe bars with transparent colors.
- Iso-contours that can be generated to highlight the area of interest
- Measurements for identifying distances between parts, different stages, points and distances

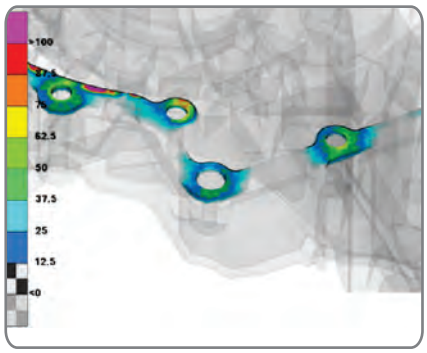


Communication of the results



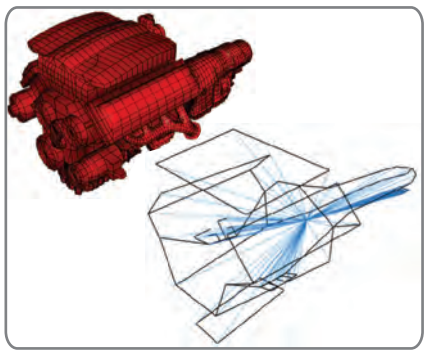
μETA integrates a very powerful functionality for the identification of areas of interest and hot-spots. The results of filters can be presented by selecting the filtered entities, using the identification tool, by isolating the filtered entities, or by adding annotations. Filters can be saved and then reused. Additionally, filters can be automatically synchronized with states and thus when moving from one stage to another the filters will be re-applied.
Results can also be presented in a tabulated manner through the Statistics tool. This list can display nodes, elements, parts and materials. It can also display user-specified results and statistic values. The user can add custom columns, create mathematical operations between existing columns and transfer data to the embedded spreadsheet editor for further process.

Parametric points



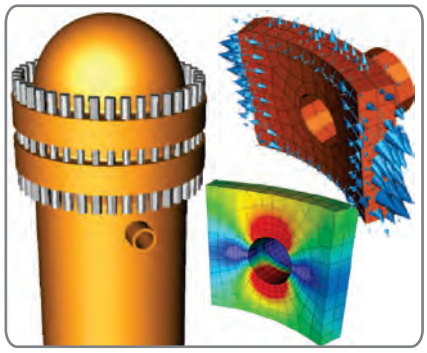
The identification of arbitrary points can be accomplished by either manually selecting them using the mouse or by specifying the coordinates, even if the points lie outside the model boundaries. Additionally, it is possible to identify points and nodes on a linear or circular path. Parametric points are available in various tools, such as the Identify and the Statistics tools.

Calculations in μETA



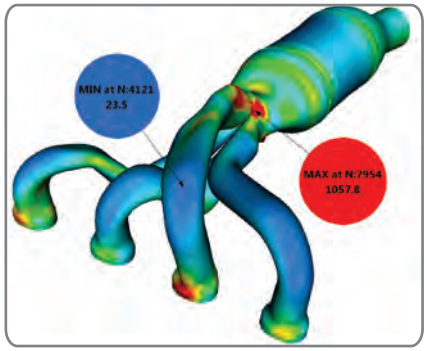
Numerous tools are available in μETA to calculate results directly based on the results read.

- The Linear Combination tool allows for the combination of results of unit loads for the generation of combined loads results
- The User Field Function tool allows for the creation of new results sets as a function of existing results. It can generate deformation, scalar and vector results
- The Modal Correlation tool allows for the calculation of the modal assurance criterion
- The Modal response tool can calculate responses from the normal modes either with loads imported through a keyword file in μETA, or directly generated in μETA
- The Modal Model Builder enables the generation of a modal, "reduced" representation of an assembly, given the results of the modal analysis



Section forces

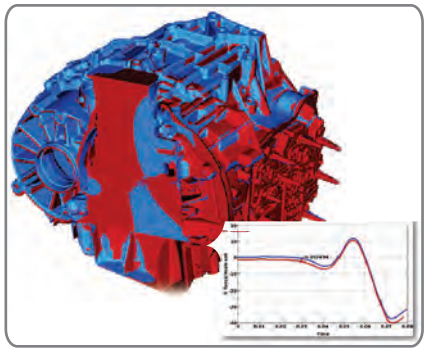
The Section forces tool of μETA is a valuable tool for the calculation of Forces and Moments on sections. Its accuracy allows for the calculation of new sections within μETA without having to repeat the task of defining them in Pre-Processor and running again the analysis. Moreover, the Section Forces tool offer the option to export grid force vectors. These can be used as Boundary conditions to accurately replace areas of the model with little or no interest. In this way the model size can be reduced while in the same time the accuracy of the calculations is improved.



Model comparison

Dedicated tools are available for the comparison of different models and different runs.

- The Multi-model statistics tool, presents in a tabulated manner a comparison of extreme values between loaded models and for the user selected states. User-specified properties, materials, groups, elements or nodes can be added to the table
- The overlay tool can use the currently generated session file or project file as a template and by adding a different set of result files, performs exactly the same post-processing actions on both 3d and 2d-data



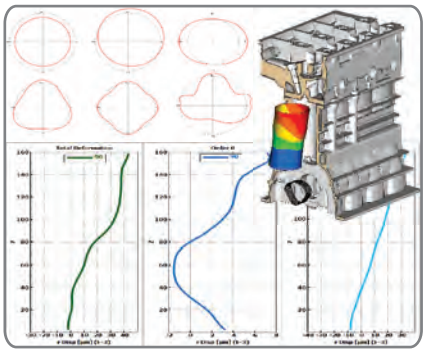
2d plots

2d plots can be generated directly from solver result files or from the 3d-model. Also, test data are supported for the correlation of simulation and real test results. Also ,2d plots can be synchronized with the 3d model.

User toolbars

µETA comes with a collection of user toolbars.

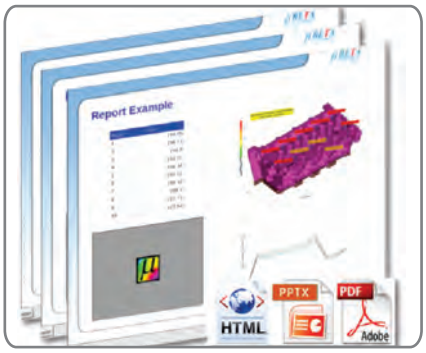
- The Cylindrical Coordinate System toolbar allows for the effortless generation of cylindrical coordinate systems, for transforming results on bores and cylinders
- The Equivalent Radiated Power (ERP) toolbar performs the calculation of equivalent radiated power results on element, part or model level and these results are visualized as new result labels. ERP results are calculated based on displacement, velocity or acceleration results output by Nastran
- The Bore distortion analysis toolbar enables the calculation of Fourier orders of deformations on bores. Selected orders can be plotted. Additionally, a new state with the 3d deformations for selected orders is generated



Courtesy of Volkswagen AG and Forschungsvereinigung Automobiltechnik

Reporting

One of the strongest points of meta-post is the very powerful report generation. Reports with custom contents and layout can be generated through the Report Composer and exported in html, pptx or pdf formats. Images and tables can be easily added, by dragging and dropping. Text boxes can be added and formatted. Basic shapes can be generated in the common PowerPoint-style. The report generation can be fully automated for a quick, ANSA deck-info like html report. The Model Report tool can be also used, for selected states and results.

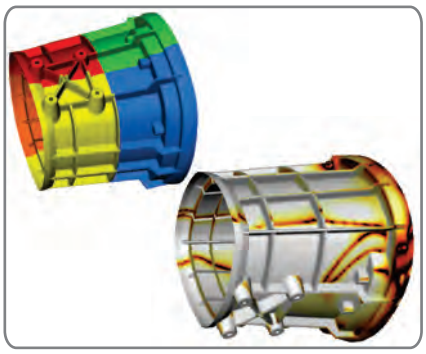


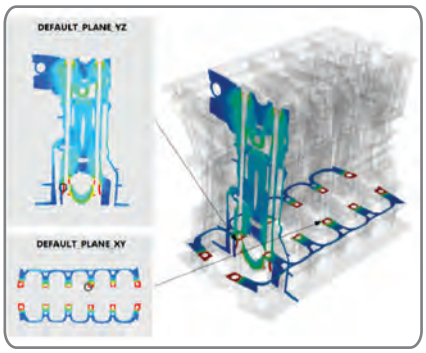
Features

- Ability to handle complex models
- A variety of tools for the assessment of interest areas and hot spots
- Statistics tool
- Ability to transfer data from the statistics table to an embedded spreadsheet editor
- Calculation of results based on solver inputs
- Powerful graph tool
- Correlation and synchronization of simulation and real test results
- Automation of post processing tasks
- Comparison between of different models and runs
- A PowerPoint-style Report Composer that can be fully automated
- Can be coupled with any parametric optimizer

Benefits

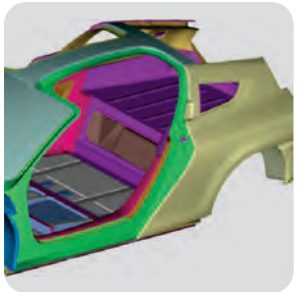
- Time-efficient post processing that minimizes cost and time to market
- Easy handling of complex models
- A wide collection of tools in an intuitive user-interface
- Flexibility through filters use
- Powerful automation capabilities allow the standardization of post processing sequences and the fast generation of reports in pptx, html or pdf formats



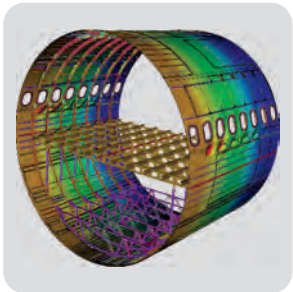


ANSA
µETA
PostProcessor
pioneering
software systems

CAE tools for products made of composite materials

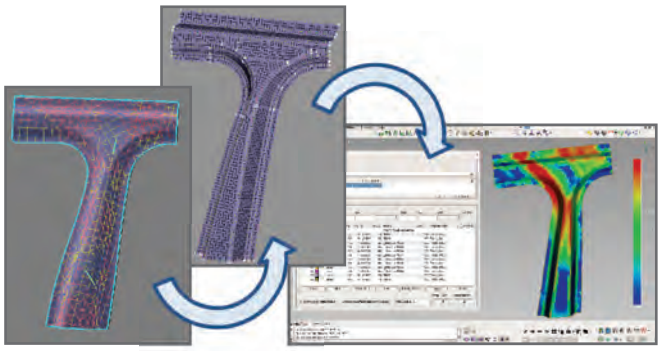


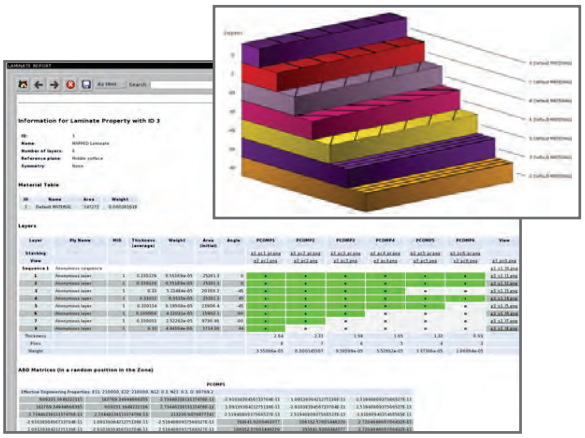

Composite materials demonstrate significant advantages such as weight saving potential, increased strength and greater design flexibility. However, these are combined with significant design restrictions and considerations. In order to fully exploit the composite materials potential, cost and time effective design, analysis and manufacturing processes are required. New software tools are offered by BETA CAE Systems for the modeling, simulation and analysis of the behaviour of products made of fibber reinforced laminated composites. The Laminates Tool and other special features were developed and integrated within ANSA pre-processor. Also the CompositePost Toolbar is offered within μETA post-processor. These tools, combined with the rest of the high performance functionality of the complete pre- and post-processing environments of ANSA and μETA software suite, offer a unique solution for the simulation of products that include parts made of composite materials.

ANSA Laminates Tool

Using the **Laminates Tool**, in combination with a collection of software features of ANSA, the analyst can master the laminates model to be submitted to the solver. ANSA enables the analyst to perform automatically, or through a graphics user interface and with the assistance of comprehensive interaction, several distinct tasks, such as:



- Bidirectional interface with CAD-based software for composites design
- Mapping of the laminates information, derived from CAD-based software, to the discipline specific finite element mesh model within ANSA, using the ANSA Results Mapping tool. The mapping procedure, leads to the lossless assignment of material orientation, material definition, element thickness per layer and element orientation per layer, attributes to the CAE meshed parts
- Building, handling and modifying all the Layers and Sequences (Groups of Layers) of a composite model, without any concern for the solver induced constraints, exploiting a special Laminate Property generated in ANSA
- Visualization, check and modification of the material orientation
- Check of the material definition
- Synchronization of material definition with available material database
- Visualization of the element thickness
- Visualization of element information
- Review of the orientation and thickness values, per layer, through a comprehensive report
- Management of the properties list
- Check of the laminates property, by acquiring report on nominal thickness and orientation of fabrics
- Generation of summaries and reports of the laminates model
- Generate and output to CAD updated layer boundaries and layer attributes



- Automatically create 3d-curves per ply for the ply boundaries and obtain them as geometric description into a file, which can be propagated back to the design
- Preview of how the laminate property will be outputted to each solver. Abaqus, Nastran, ANSYS, LS-DYNA, PAM-Crash, Radioss and PERMAS, solvers are supported
- The final laminates definitions, based on the FEA model as is prepared to be submitted to the solver, can be outputted and fed back to the CAD-based composites design software

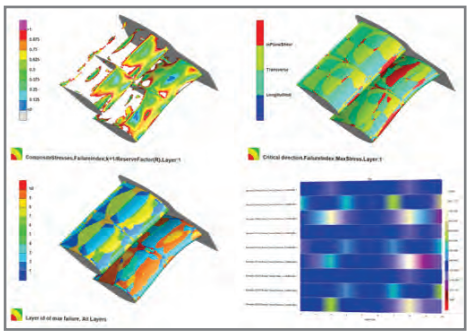
μETA CompositePost Toolbar

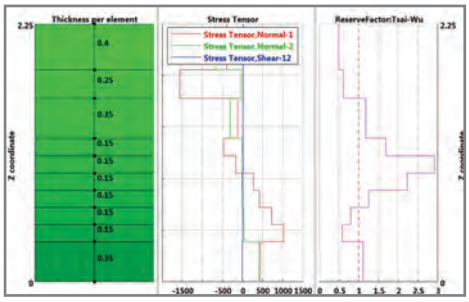
Analysts demand to evaluate results of intricate products and complex simulation models in a high-performance manner, with direct references to the initial materials of the design process. Therefore, special consideration is given to the provision of automated tools for the post-processing of results deriving from the analysis of products from laminated composites.

The solver results from the analysis of products made of composite materials can be processed by μETA post-processor for evaluation, report generation and decision making towards the design improvement. In order to accommodate the requirements of the analysis of these results, μETA offers the **CompositePost Toolbar**, a set of integrated specialized functionalities, collected into a single user interface.

This collection of functions offers:

- Identification of layers as they were defined in pre-processing.
- Calculation of Failure criteria, Reserve Factor and Effort (Tsai-Wu, Puck, etc.)
- Identification of the layer with the max failure
- Identification of critical directions
- Identification and isolation of failed elements
- Plot of stress tensor results across each element thickness
- Re-evaluation of failure criteria after modifying material properties
- Composites results support for Abaqus, Nastran, PAM-Crash, Radioss and PERMAS

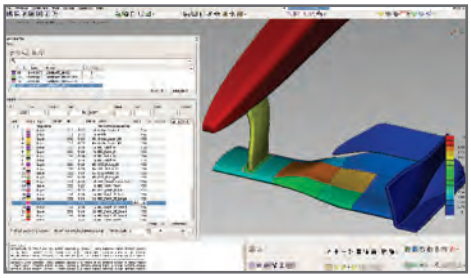


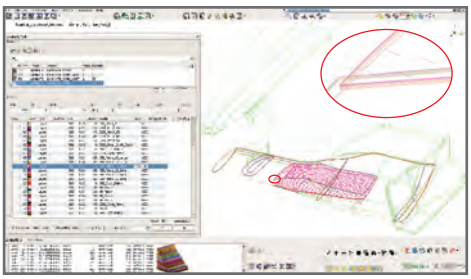


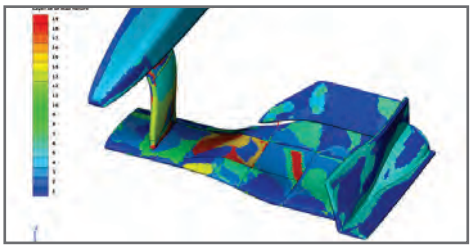
This set of functions is integrated to the complete post-processing environment of μETA, therefore, the analyst can end up with an automatically generated report to be used as a decision making tool towards the improvement of the product design.

Benefits

- Tight interfacing with CAD-based composites design tools, so analysts access the same CAD master model of a composite part in its to-be-manufactured state
- Two-way interaction between designers and analysts, allows the multiple iteration of design changes
- Solver & discipline neutral modeling removes all solver induced restrictions
- Parallelized modeling for numerous solvers
- Deal with composite parts properties and detailed composite lay-up descriptions, with the level of abstraction required by each analysis type
- Handle full-model built-up of models comprised of a large number of parts, of a variety of materials and properties, and joined by numerous connections of different type
- Building and handling of model variants
- Monitoring of model updates
- On the fly handling of Zone and Ply definitions
- Propagation of design changes and updated ply boundaries back to the CAD-based design tools
- Automated calculations, graphs and report generation based on analysis results
- Comprehensive identification of layers and propagation of the identification information throughout the process
- Results evaluation with direct references to the initial materials of the design process
- High level of automation and customization
- Incorporation of laminates design into optimization cycle
- Designers and analysts still focus on their domain of expertise, using their own software tools

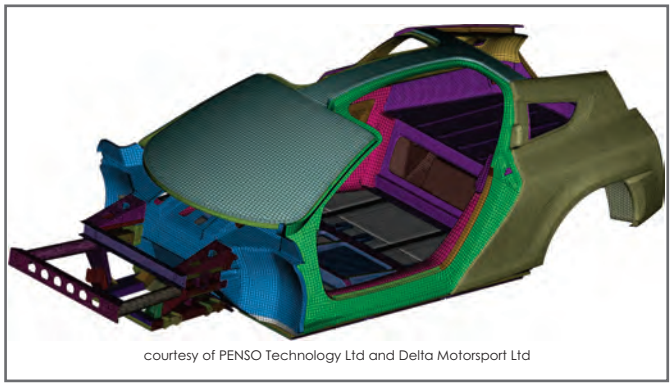






Full-product simulation

ANSA pre-processor and μETA post-processor software suite has a proven record of complete CAE model buildup and analysis results handling. The software meets the Industry's requirements and exceeds legacy procedures expectations, due to the level of automation and customization they offer. The implementation of laminates modeling and analysis functionality into ANSA and μETA, with full product model handling capabilities, enables analysts to:

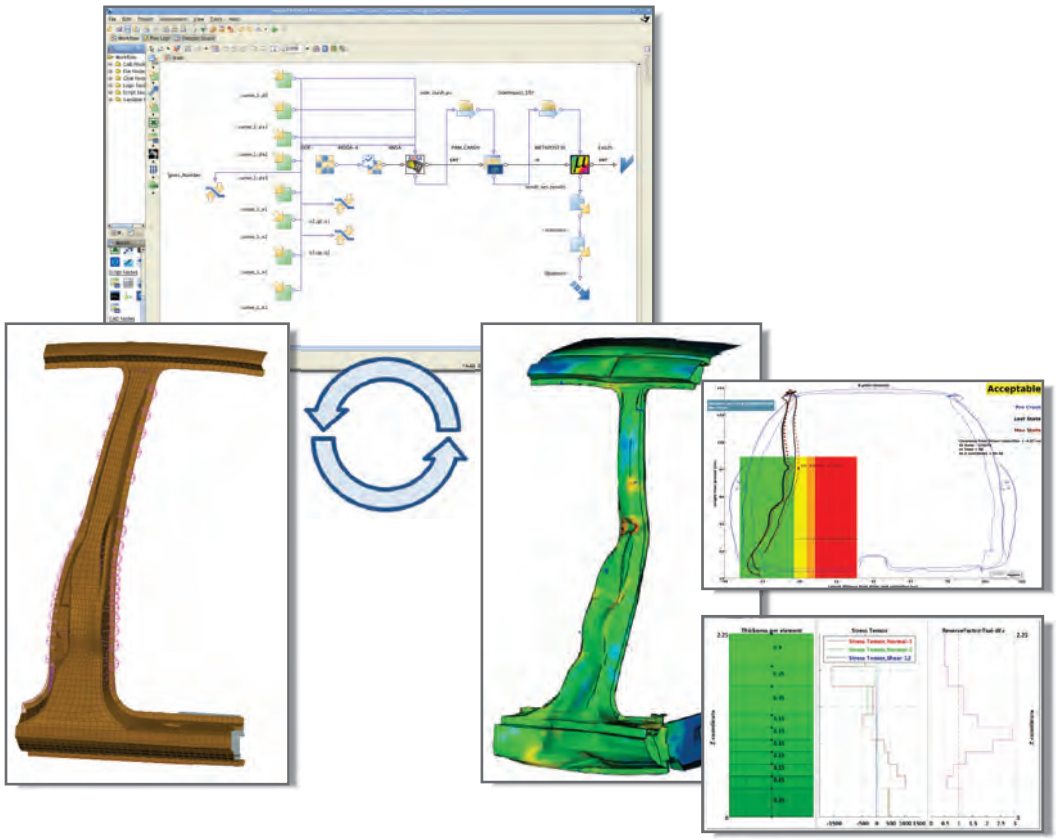


courtesy of PENSO Technology Ltd and Delta Motorsport Ltd

- build and handle model variants,
- handle efficiently the model updates,
- assemble models using a large number of standard and customizable connection types, and
- perform multidisciplinary modeling and processing of analysis results for numerous FEA solvers and analysis types.

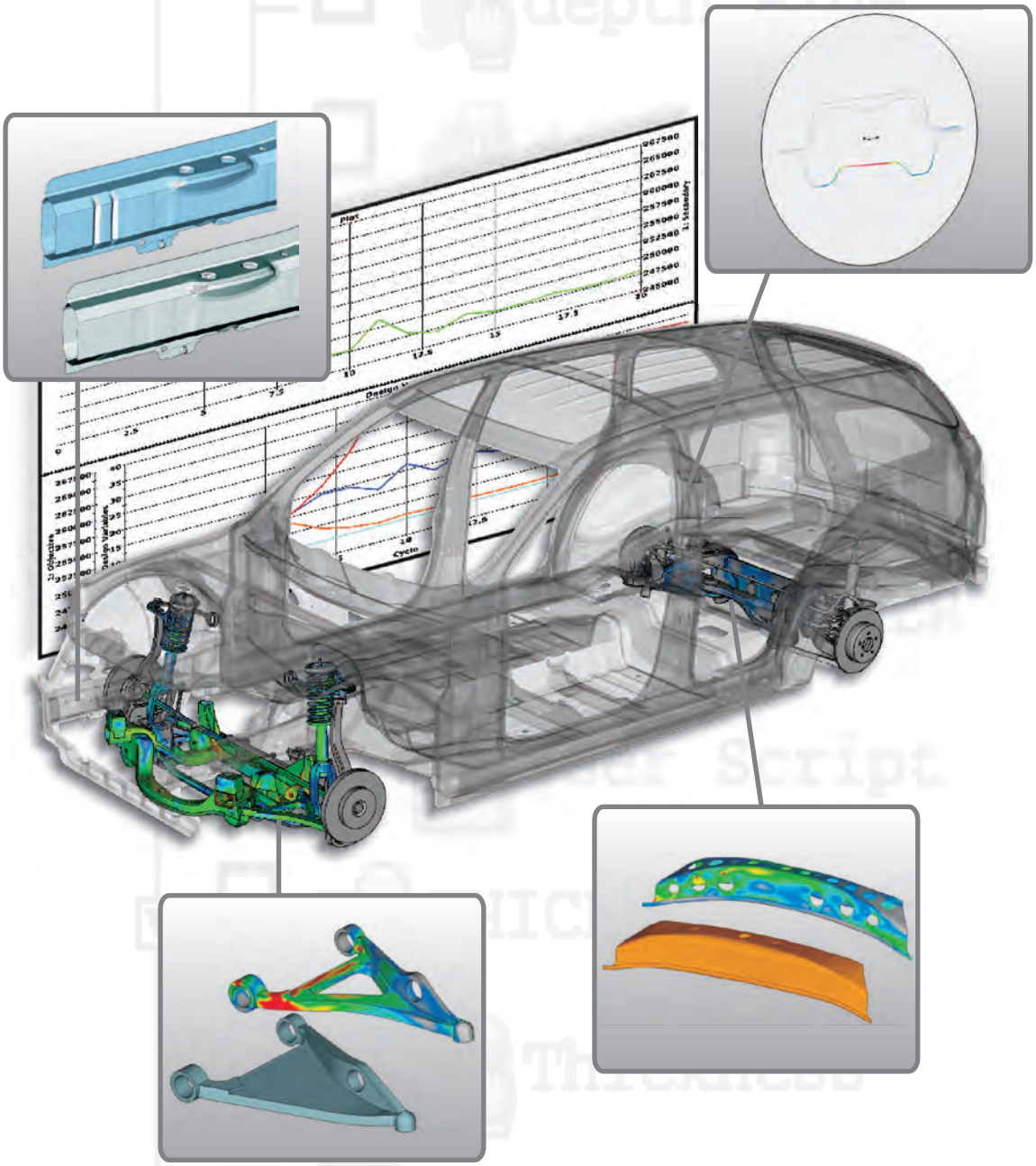
Optimization

The capability of ANSA and μETA to be coupled with numerous optimizers and the ability to return the design and properties changes to the CAD-based composites design tools, allow the design of products, which include parts made of composite materials, to be easily optimized.



ANSA
µETA
PostProcessor
pioneering
software systems

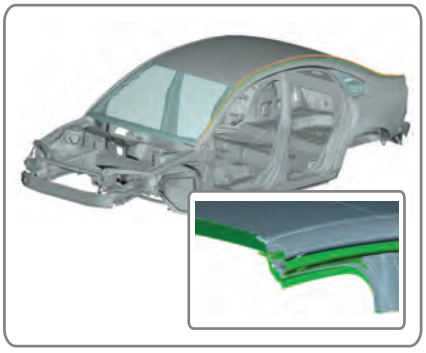
prerequisite for effective optimization



ANSA pre-processor and μETA post-processor in combination with all popular optimization codes, provide a complete tool for optimization applications.
From concept design to final testing, ANSA & μETA package brings enormous performance and versatility to the optimization problem set-up.
The ability to control the model shape using the ANSA Morphing Tool, ANSA model values and even complicated tasks such as batch meshing and model checking, makes the tool unique.

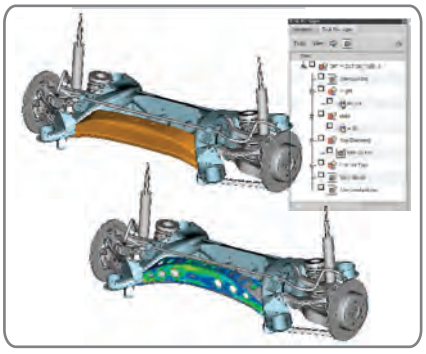
Morphing features

- Shaping of FE model through the use of 3D, 2D, 1D and Cylindrical boxes
- Morphing on CAD geometry
- Morphing Parameters to control model shaping in a parametric way
- Parameterizing any manual morphing operation by the DEFORMATION parameter
- Accurate morphing of model feature lines by fitting on target curves
- Recording of morphing states for easy recovering of any previous shape
- Automatic reconstruction to improve mesh quality after morphing
- Special entities, called Nested Elements, to constrain the shaping of specific features of the model such as holes and beads



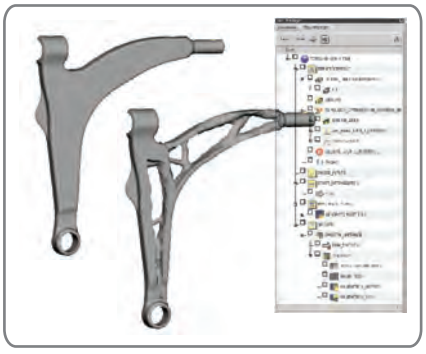
Direct Morphing

- A powerful new morphing algorithm performs direct morphing on geometry or FE model without the need of morphing boxes. This process minimizes the set up time especially in large models
- Direct morphing can be applied by means of edge fitting or Control Points movement
- Depressions and simple features definition
- Parametric control of holes diameter
- Cross section morphing



Process automation

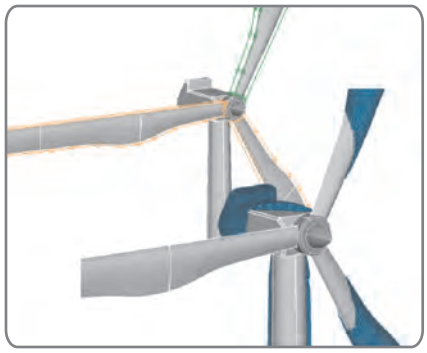
The integrated process automation tool facilitates the set up of the optimization sequence. Design variables are defined in the Optimization Task and are connected with any Morphing Parameter and thus control the shape modifications of the model. In a similar manner Design variables are connected and control any parameter of any ANSA entity (shell thickness, material density, spotweld distance, etc.). Furthermore, complicated actions, such as features creation and treatment, parts replacement and mesh quality improvement are assigned to the process and driven by design variables.



Coupling with optimizers

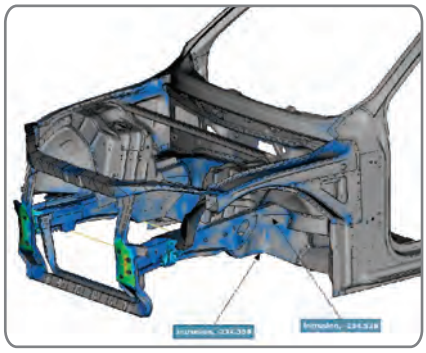
- Direct coupling with LS-OPT, modeFRONTIER and OPTIMUS, without the need of any scripting
- Coupling the Optimization Task with any other parametric optimizer, such as Isight and DAKOTA, without the further need of scripting
- Integrated TOSCA Structure interface which is able to define Topology, Shape and Bead optimization scenarios
- Automatic meshing quality improvement of the optimum model and automatic definition of the validation model after TOSCA Run
- Monitoring the TOSCA Run through the ANSA Interface

Simulation and model validation



- A special tool to simulate and animate the model shape for different combinations of design variable values gives the user the ability to check the model before contacting with the optimizer
- Video recording of the animated model
- Enhanced Design Of Experiments, Full Factorial Algorithm for easy definition of experiments
- Definition of full model reports during the optimization process checking model validity
- Automatic model fixing, such as property thickness penetration, is performed during the optimization loops

Spotweld optimization



- Parametrization of weldings and easy handling by the Optimization Task
- Control parameters such as spotweld distance, number of spotwelds, spotwelds' diameter, connections' properties or materials, and alternative connections representation types
- Combined spotweld and Shape Optimization Morphing Tool can handle Connections Points, Lines and Faces in the same way as the Morphing Tool can handle Connections Points, Lines and Faces
- Distribution and control of spotwelds by defining connection density function along the connection line

Features

- FE & geometry morphing 2D & 3D parametric morphing
- Edge fitting
- Direct morphing
- Process automation
- Scripting

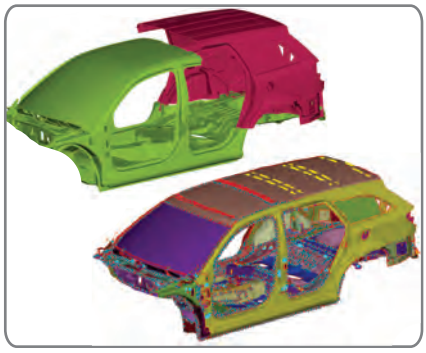
Set up for:

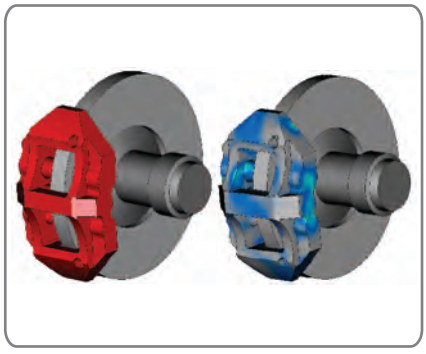
- Shape and Parameter optimization
- Composite material optimization
- Spotweld optimization
- Multidisciplinary optimization

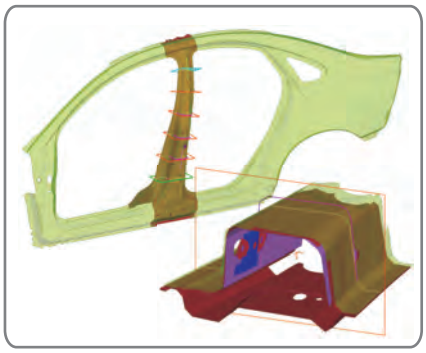
- Coupling with numerous optimizers
- ANSA & μETA nodes in ModeFrontier and OPTIMUS interface
- LS-OPT direct interface
- TOSCA Structure integrated interface
- NASTRAN SOL 200 interface
- μETA post-processing for optimization

Benefits

- Minimizes the set up time in morphing using the Direct Morphing functionality
- Powerful morphing in CAD geometry
- Common approach in coupling with any optimizer
- Fast and flexible optimization sequence set up using the massive definition of design variables





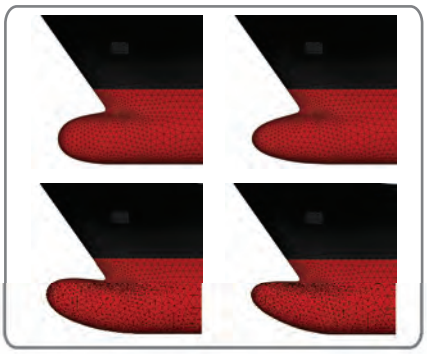


Multidisciplinary optimization

- A defined optimization sequence can be applied on different representations of the same model and prepared for different solvers and analysis scenarios for the set up of multidisciplinary optimization problems
- The use of common Optimization Task and Morphing Boxes in different analyses ensures the identical shaping

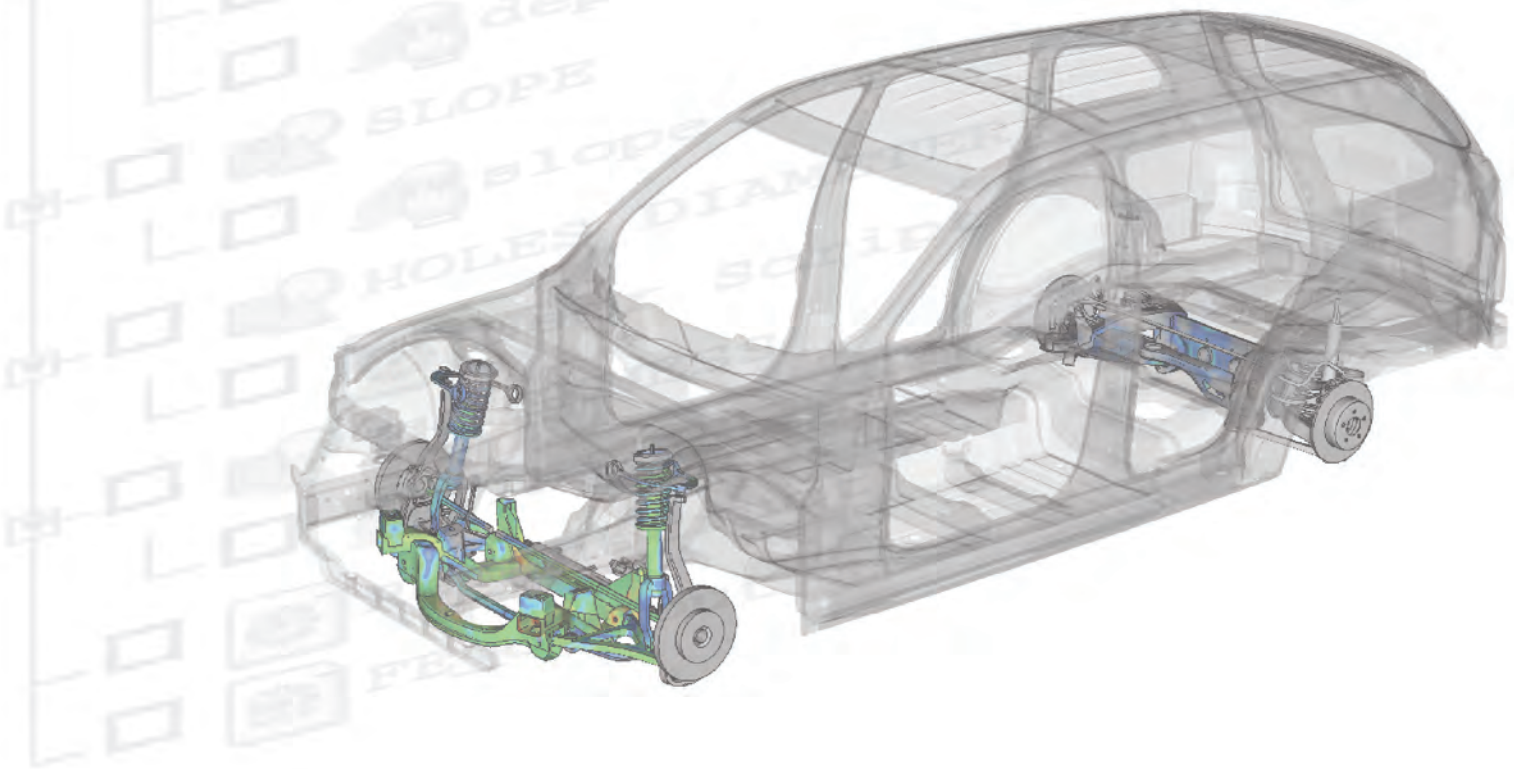
Post-processing

- A special tool in µETA offers a flexible way for the responses and histories extraction, from the solvers result files
- Responses extraction from the 3D model and the 2D Plot
- The automatic definition of postprocessing sessions to participate in the optimization loop, and the calculations upon solvers results, are only a few of µETA's powerful capabilities



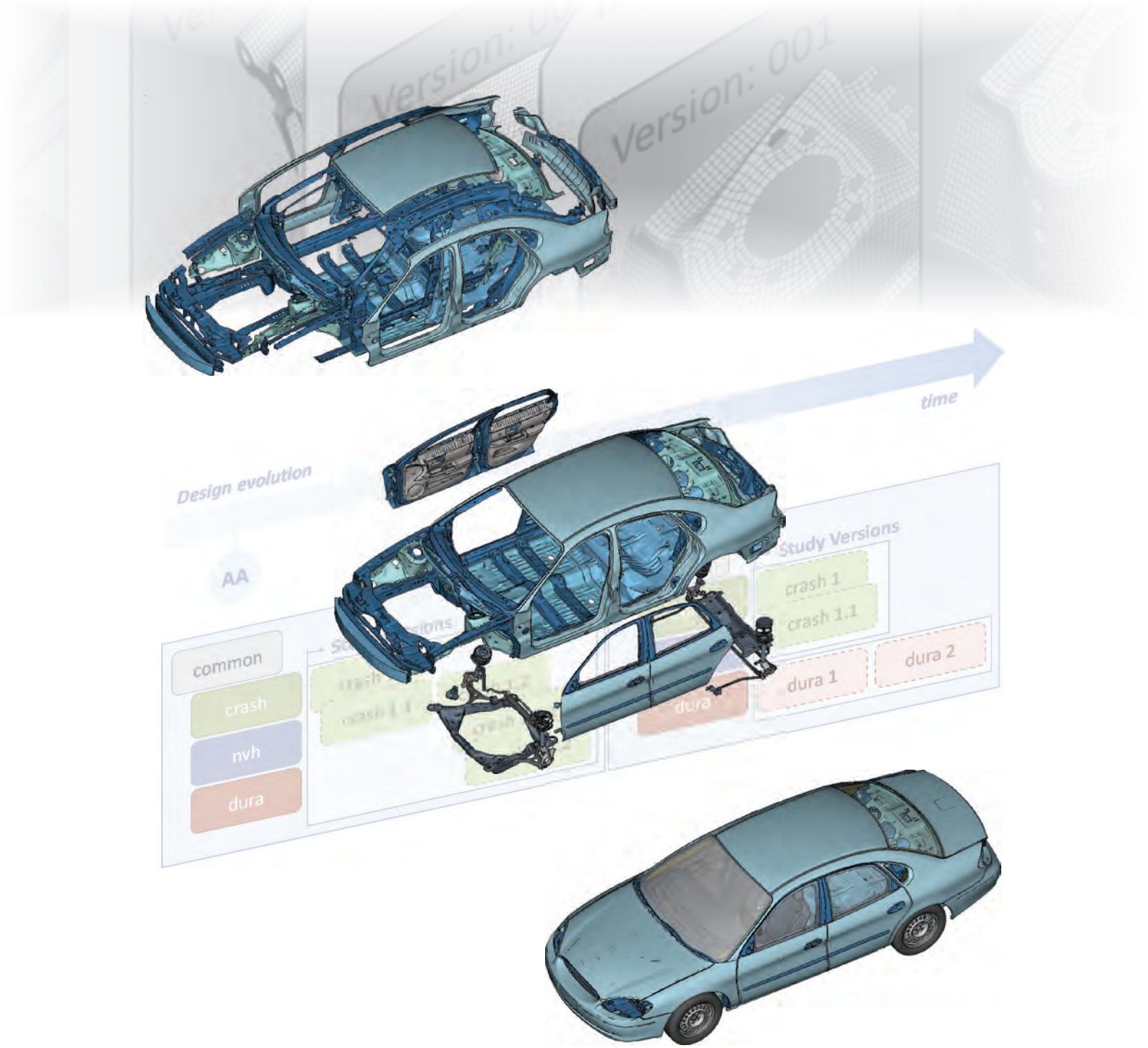
NASTRAN SOL 200

- Support of the required keywords for the definition of NASTRAN SOL 200 cases for property, and material optimization
- Use of the Morphing tool for the massive definition of the DVGRIDs for shape optimization according to the Manual Grid Variation method
- Support of the keywords TOPVAR, TOMVAR and BEADVAR for topology, topometry and topography optimization



ANSA
pioneering
software systems

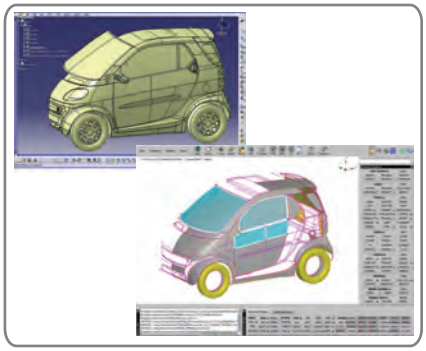
f a c i l i t a t i n g
Data Management in ANSA



Data Management is unquestionably one of the most critical factors that contribute to the efficiency and productivity of CAE teams. The huge amount of data involved in pre-processing as well as the need for synchronization of the CAE model with the design evolution, intensify the need for flexible and inexpensive solutions that can bridge the gap between the PDM systems and the CAE-world and become a reference point throughout the CAE cycle.
Addressing these needs, BETA CAE Systems provides integrated solutions for the effective and cost-efficient management of data, starting from the PDM systems export and all the way to the output of the keyword file. Out of the box solutions for the interaction with PDM systems and breakthrough CAD input technologies ensure a smooth and effortless transition to the CAE world. From then on, the ANSA Data Management (ANSA DM) takes over, streamlining data exchanges between engineering teams and assisting the engineers in the model management on both ANSA- and Include- files basis.

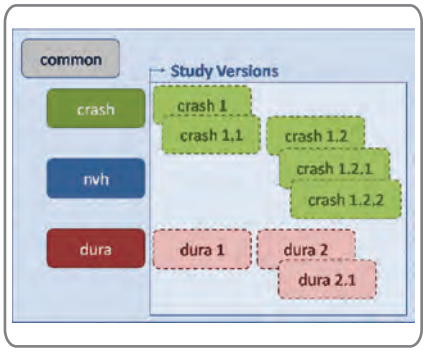
Interacting with PDM/PLM Systems

CAE processes begin by gathering information related to the simulations that will follow. The bulk of such information consists of the vehicle product structure and the related CAD- and meta- data, but may also extend to the loadcase scenarios, the analysis results and the respective evaluation reports. As this information is usually handled by an enterprise PDM/PLM system, ANSA is able to communicate with established PDM/PLM platforms (e.g. Siemens PLM Teamcenter, Simulia SLM etc) to collect the required information and feed it to the ANSA Data Management system, in order to serve downstream CAE processes.



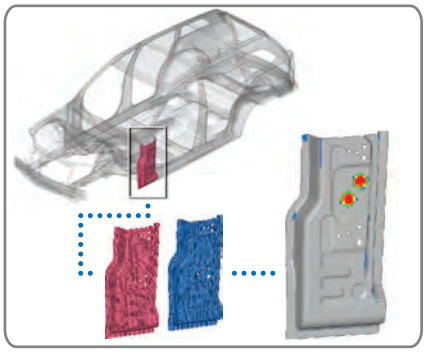
CAD Input Technologies

The conversion of CAD data into ANSA files is one of the important starting points of the CAE workflow. Therefore, the CAD-Translators aim at the production of a high quality geometry result, providing the automatic tools for the preparation of the CAE "representation" of the components namely, automatic cleanup, mid surface generation, thickness and material assignment, interpretation of CAD/PDM-attributes (such as CAD version, release date and designer's name). The conversion phase is not limited to the geometric definitions, but it also expands to the extraction of assembly information like hierarchy and positioning data, as well as to the conversion of connection information (such as spot welds, seam welds, adhesives, etc.). The CAD input is supported for the following formats: CATIA, CGR, NX, JT, Pro/ENGINEER, SolidWorks, Invertor, IGES, STEP, VDA.



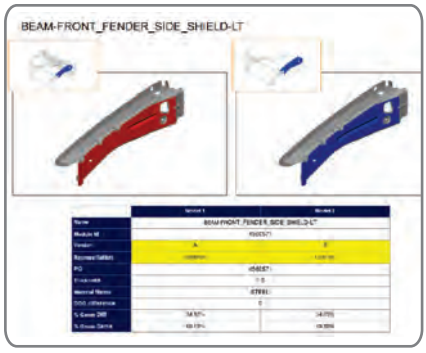
Representations Management

Different representations can be created and stored, facilitating the use of a component in multiple disciplines. Serving this purpose, representations can either be detailed or reduced FE-models. All the detailed FE-representations of a component are created by the Batch Meshing tool on a common geometrical basis. Reduced representations, like the lumped mass, are abstractions of a detailed representation, suited for a particular analysis. The Parts Representation Manager controls the generation of new representations and the direct switch from one to the other. Each representation can accept an arbitrary number of study versions, allowing the introduction of design changes on FE-model level.



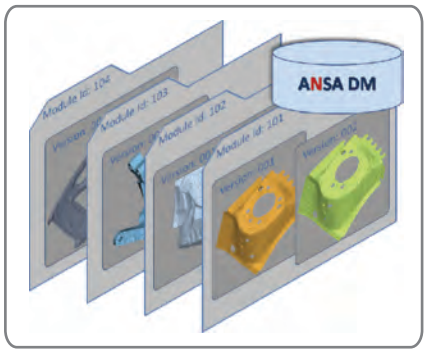
Compare

To assist and accelerate decision making, the Compare tool allows for the fast identification of differences in geometry, connections, and solver-specific definitions. From a single part to a full scale assembly, the organization of information, the easy navigation and the synchronization of the comparison report with the drawing area make the compare tool ideal not only for tracking changes but also for selective model updating, enabling the transfer of model attributes to the model at hand.



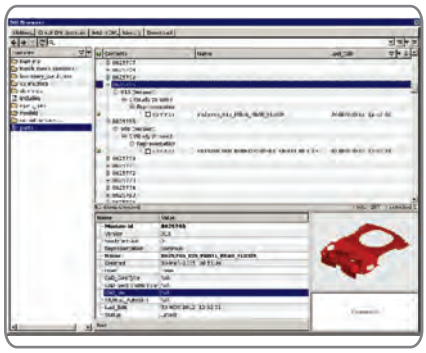
Updates notifications

ANSA DM makes the timely identification of component updates possible by monitoring all changes related to the model at hand. Newer CAD versions, study versions or plain file changes can be detected and, with the aid of the compare tool, engineers are able to decide whether the model should be updated or not. Identified updates are incorporated in the model by direct replacement of the respective older versions currently in use. During this process, all affected connection and mass information, as well as boundary conditions are automatically adapted to the model changes.



DM Browsing

The efficiency of any data management solution is dependent upon the ease with which the right data can be found. The DM Browser enables the identification of ANSA and Include files using predefined filters or user-defined queries. The creation and last-edit dates, the user name and the user comments are only a few of the file attributes that can be "scanned" by DM Browser. The results of a querie can be directly merged in the model or replace their variants currently in use.

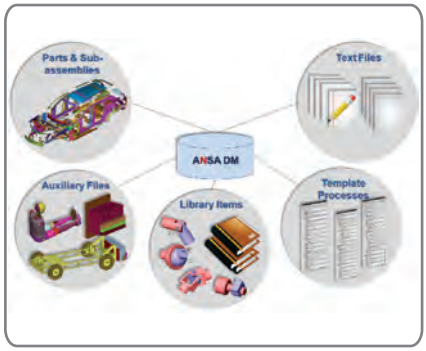


Features

- Direct input of product structure from CAD/PDM
- Direct open of CAD files in ANSA
- Multiple-instances recognition and special treatment
- Parts, groups and include files caching
- Study versions organization
- Check for DM Updates
- Compare tool
- Model variants handling
- Browsing based on queries

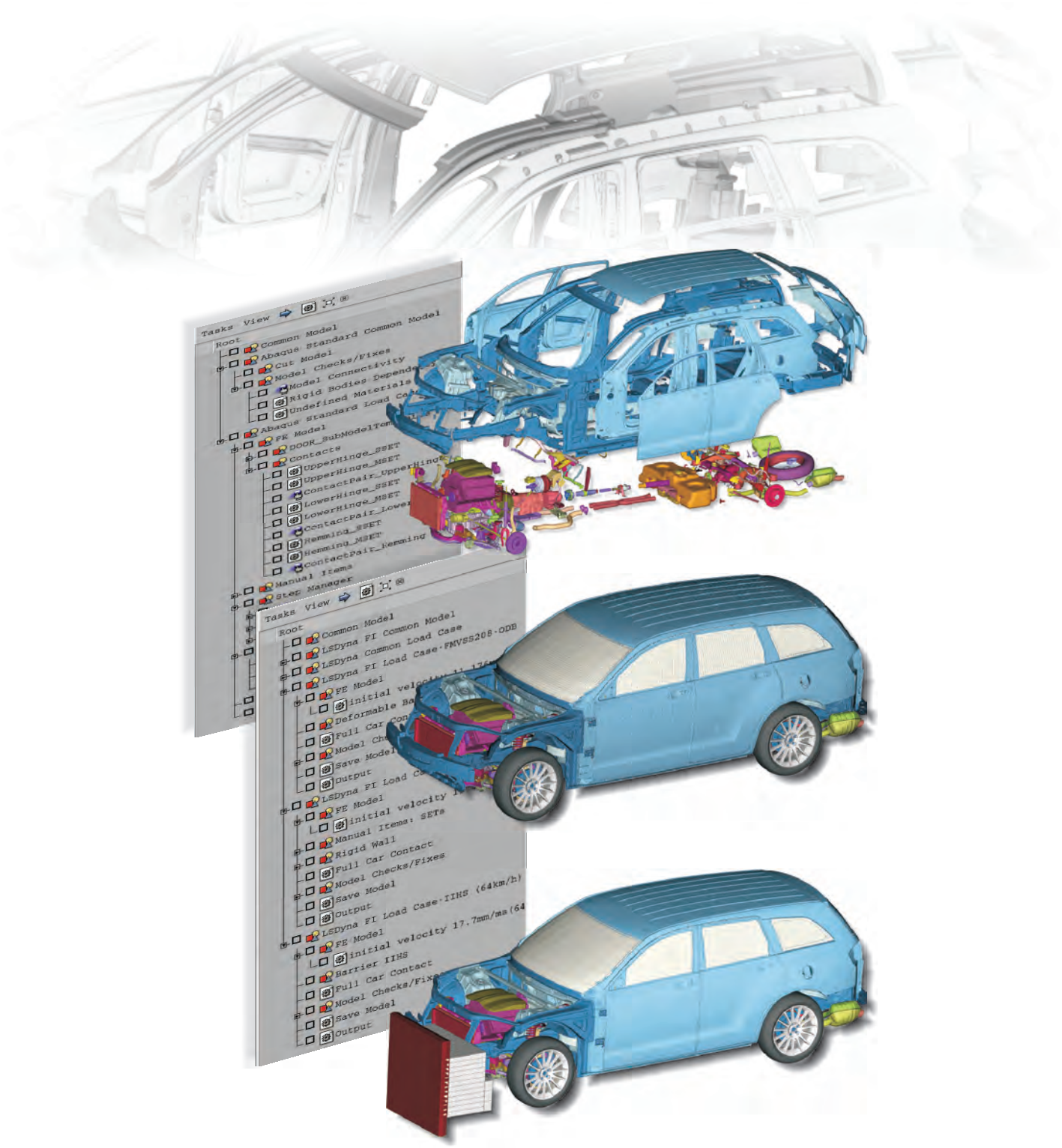
Benefits

- Direct generation of connections from CAD data
- Flawless translation of native CAD files into ANSA files
- Organization of all pre-processing data
- Maintenance of CAD/PDM meta-data in CAE model
- Synchronization of the CAE model with CAD evolution
- Collaboration among engineers and departments
- Assistance in decision making
- Adaptivity to existing practices
- Automatic updates notifications

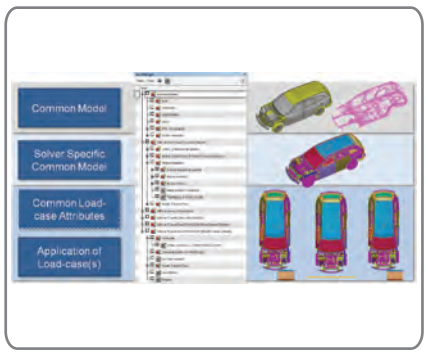


ANSA
µETA
PostProcessor
pioneering
software systems

pioneer solutions for process automation

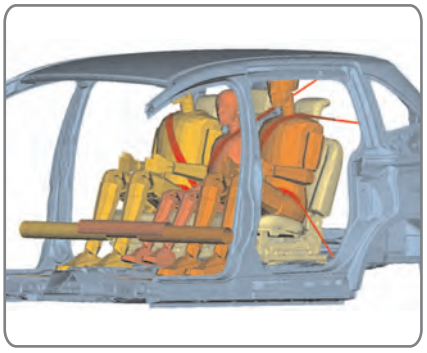


Today's CAE cycles involve a great number of standardized processes which, given a certain input, must produce a result that meets a set of requirements. Process automation capabilities of ANSA can accurately capture any pre-processing task making any quality criteria and requirements inherent to the process. As a result, the productivity of engineering teams is improved and the quality of models is assured, leading to a drastic reduction of the CAE turnaround time.



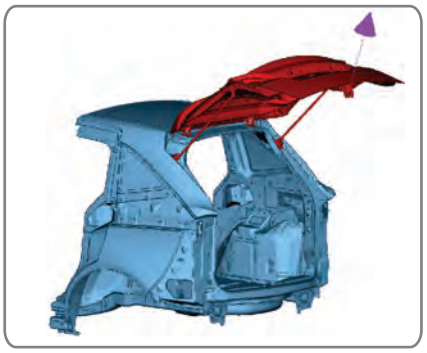
Task manager

At the heart of the integrated process automation lies the Task Manager, a tool where all individual steps of the development of a simulation model are included in the form of Task items. Tasks in Task Manager are built up by analysis experts and reflect all the distinct actions that must be performed for the set-up of a high-quality simulation model. Once a Task is completed, it can be saved as a template process and can then be used by inexperienced users to generate the required analysis models, promoting knowledge transfer within the organization.
Tasks are not bound to any particular set of data, thereby enabling the repeating use for similar analyses on different models. Additionally, advanced tools such as the Parts Representation Manager, Batch Mesh Manager and Connection Manager are orchestrated by Task Manager, forming seamless processes that cover the complete pre-processing workflow. Task Manager is capable of tracking dependencies between modeling actions, assuring that, whenever a change takes place in model parameters, all implicitly related definitions are properly updated.



Scripting

The BETA Scripting Interface is an Application Programming Interface that is an extension of the Python programming language. An ANSA script is a Python script. It can be used to extend the core functionality of ANSA. It can automate procedures of CAD data translation, meshing, boundary condition definition and model output. A rich library of functions enables the creation and manipulation of model data. Custom graphical user interfaces can be generated with the aid of a Graphical User Interface designer. Additionally scripting can be used to generate a big number of capabilities such as custom connection models and tailor-made model checks. The Python Programming language allows the use of numerous external libraries (i.e. mathematical, scientific) which offer great capabilities in the hands of the user.



Features

- Python programming language
- Customization of core functionality
- Complete process capturing
- Process templates
- Customizable Graphical User Interface

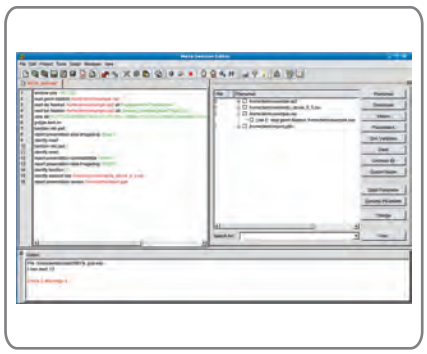
Benefits

- Process standardization
- Action-dependency tracking
- Model quality assurance
- Automatic model adaptation to changes
- Reusability
- Repeatability

µETA provides the analyst with a unique range of powerful tools to set up automated 3D and 2D post processing tasks. Special action based language (session), Python programming and user defined toolbars can boost productivity offering post-time reduction, limitless automation capabilities and reliable repeatability.
Manual post-processing is recorded, parametrized and executed through intuitive GUI without the need for text editing. New processes can be built up inside editor tools with specifically developed functionality for online help and debugging.
Toolbars can be shared on group or company level while processes on local data can easily be packaged and transferred.

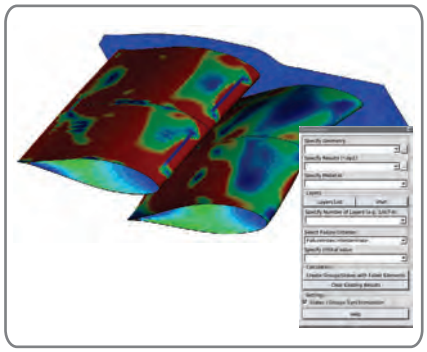
Session files

µETA is a GUI operated software that also supports and assigns commands to all actions, so that everything can be recorded in text files and be reapplied sequentially without the need for user interference. The Session Editor, a tool integrated in the software, facilitates the creation, editing and parametrization of session files through error checking, coloring of the command arguments, identification & listing of argument types (e.g. filenames), and debugging. Furthermore, sessions with local data used by this can be packed together in order to enable their easy transportation. Finally, sessions can also be executed in batch mode without the need to have GUI.



User toolbars

Better organization of work and time efficient post-processing can be achieved through the use of toolbars. All necessary analysis-oriented commands can be gathered inside a single interface and assigned to intuitive gui entities including buttons, textboxes, radio buttons and many more. A tool is available for the interactive creation and designing of toolbars, whereas storage location can be controlled in order to manipulate the level of common use.

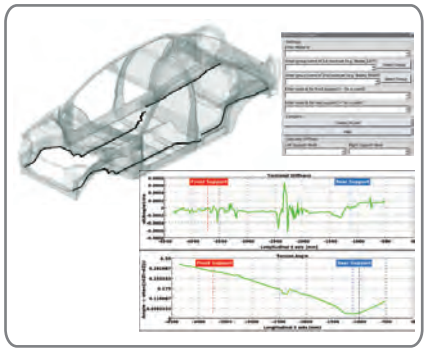


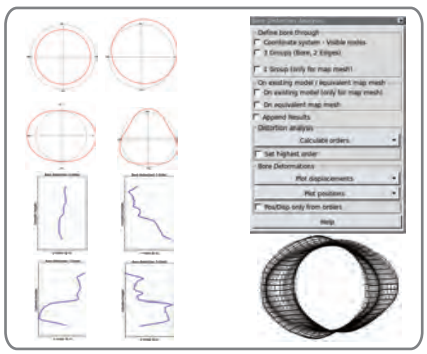
Features

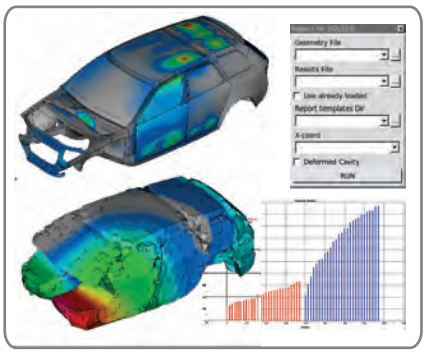
- Automation of 3D & 2D post processing tasks
- Easy set up & parameterization
- Editors for the easy creation of sessions / scripts
- Custom GUI
- Direct access to all simulation results
- Full reporting automation
- Sharing of processes among group/company level
- Stiffness calculation toolbar
- Pedestrian toolbar
- Bore Distortion Analysis toolbar
- Optimizer Setup toolbar
- Laminates toolbar

Benefits

- Time-efficient post processing
- Effortless realization of frequent tasks
- Fast model comparison
- Feasibility of advanced procedures
- Automatic results reporting
- minimizes cost and time to market
- novel features lead to results faster, while ensuring efficiency and quality.



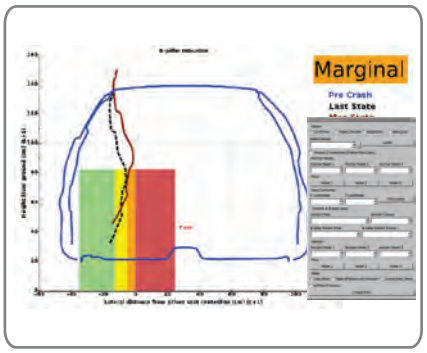




Script files

In order to automate more advanced and complicated procedures that cannot be achieved through session files, script files -text files with the common in ANSA and μETA Python programming language- can be used. The capabilities of scripting are practically limitless, as everything inside μETA can be accessed, used and modified and complex programming structures can be set up. Apart from single script execution, the user can call a script inside μETA tools, whenever an advanced manipulation of data is needed, for example inside an annotation's text, in a 2D plot legend, in mathematical operations or in the statistics tool.
Further script features include the reading/writing of ASCII and binary files, the direct session commands incorporation, and the extensive interface customization for data input.

The Script Editor, a tool developed for the creation and editing of script files, also provides error checking, debugging functionality and a full library of script functions with online help and ready to run examples.

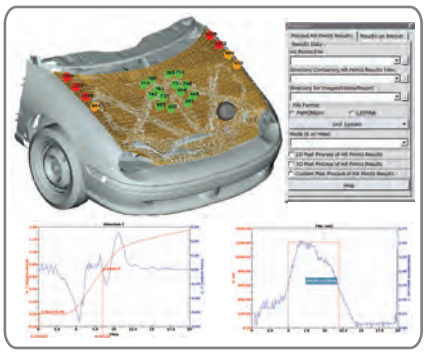


Variables and built-in functions

μETA includes tools in order to aid the easy retrieval and storage of values or other data. In the Variables tool the user can define and manipulate all available variables, static or dynamic, system or user defined, whereas in the Built in functions tool data from all entities can be retrieved through an intuitive GUI. Moreover, inherent μETA variables are automatically created to pinpoint significant session states.

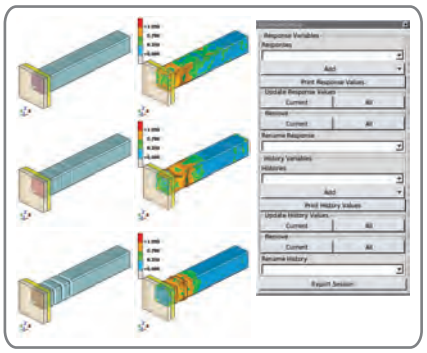
Automatic procedures' execution

In μETA the user can define some actions to be applied at each change of state/step/subase.
Furthermore, when the advanced filter tool is used inside a focus command, an identification or an annotation, the tool can be locked so that the filters are reactivated every time the state changes and the respective action is applied on the new filtered entities.



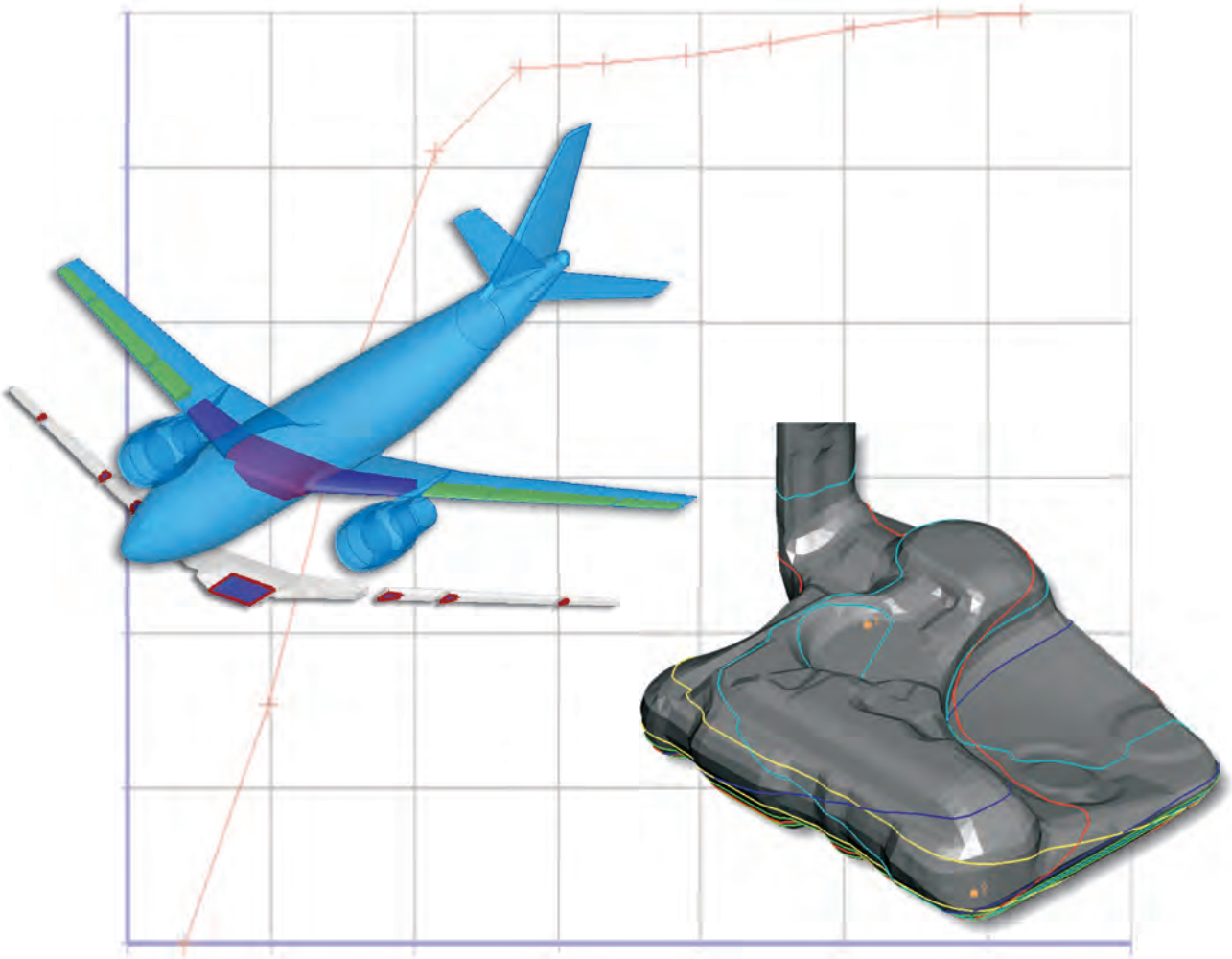
Tools associativity

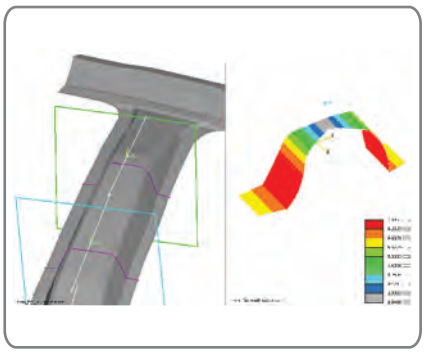
The interoperability among μETA's major automation tools can offer users great flexibility. Session parts can be combined with script parts, sessions and scripts can be called from other sessions or scripts. The session editor can be called to edit commands behind a user toolbar button. A user toolbar can be created directly from the session editor. Script code can be assigned directly to a toolbar entity.



ANSA
μETA
PostProcessor
pioneering
software systems

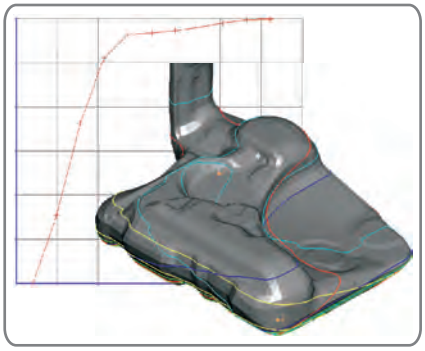
special tools





Cross-Section analysis tool

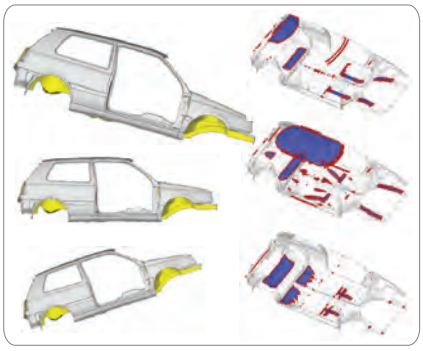
Some of the available features are:
- Cross-section definition, check and modification integrated with the main menus' functions
- Computation of Cross-section geometrical results
- Cross-section Stress Analysis for variable loading points, loads and properties
- Automatic bar or beam property entity definition, integrated into the preprocessing Decks
- Capability to create the equivalent box-cross section
- Possibility to modify a cross section using the morphing tools
- Direct editing and support of the NASTRAN keywords PBMSECT and PBRSECT
- Automatic creation of the required NASTRAN SOL200 keywords, for cross section optimization purposes
- A Group library with predefined sections, such as circular, rectangular and tube
- Cross section edit card that can be used by script functions



Fuel Tank analysis tool

Rapid simulation of the filling or emptying process of a closed tank predicting:
- Total volume of the tank and wetted surface
- Liquid levels and their corresponding volume
- Liquid and total Centre of Gravity monitoring
- 3D curve representation of levels and 2D filling curve at various user defined inclinations and filling/suction points

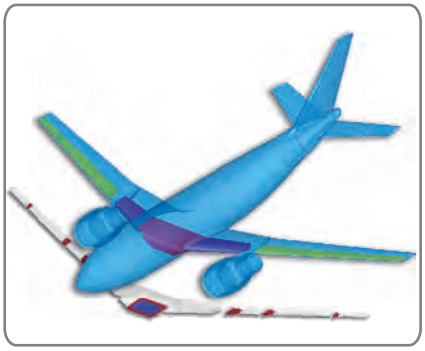
Some of the available features are:
- Volume vs fuel level graphic and numerical analysis, for arbitrary car positions
- Automatic volume-traps detection
- Filling and suction points definition and analysis
- Real filling process monitoring



Volume Traps Tool

This Tool has various applications, among which the detection of trapped air or liquid in the paint bath process of BiW assemblies, or the identification of unused fuel or settled fuel residues in fuel tanks.

BiW paint bath simulations
- Allows the identification of volume traps at a BiW that is going in and out of a bath
- Position the meshed parts like being immersed, or extracted from a virtual bath
- Get the location, the total volume and area of the air traps and liquid ponds



Unused fuel prediction
Calculation of location and volume of residual resting water or unused fuel in aircraft tanks based on:
- different tank configurations
- suction pump locations
- aircraft inclinations

notes

BETA CAE Systems S.A., with headquarters in Thessaloniki, Greece, is a private engineering software company specialized in the development of state-of-the-art CAE pre- and post-processing systems. The company, founded in 1999, incorporates research, development and business dating back in the late '80s.
The company, focusing on not only meeting but also on exceeding customers requirements, is committed to its mission to be the leading force in the CAE software field.

The flagship product package, **ANSA** pre-processor and **μETA** post-processor, holds a leading position in the CAE software market worldwide. For years now this product sets the standards in CAE pre- and post-processing in many sectors, including: automotive, rail vehicles, aerospace, motorsports, maritime design, processes engineering, energy, defense, heavy machinery, electronics, power tools, biomedical engineering and academic.
The software accommodates all features required for the pre- and post-processing for many disciplines, such as Durability, Crash & Safety, NVH and CFD.

ANSA

Is an advanced multidisciplinary CAE pre-processing tool that provides all the necessary functionality for full-model build up, from CAD data to ready-to-run solver input file, in a single integrated environment.
Some of the key CAE pre-processing tasks that can be achieved, are: geometry clean up & CAD reconstruction, shell & volume elements generation, batch meshing, parts' assembly & welding modeling, pre-processing analyses setup for all major FEA & CFD solvers, morphing, coupling with optimizers, and cross-section, fuel tank & volume traps analyses.
ANSA owes its success to a number of innovative concepts and ways of implementation such as the customizable GUI, the integrated process automation, the direct associativity of geometry to mesh, the versatile concept of meshing areas, the in-house developed fast meshing algorithms, the flexible use of the virtual connections for the definition of alternative connections' models, the concept of the Common Digital Model, the automated features for the update and modification of FE-models, the interoperability between alternative pre-processing decks, and the fast algorithms for the model's quality & integrity check and improvement.

μETA

Is a thriving multi-purpose post-processor meeting diverse needs from various CAE disciplines. It incorporates innovative features, user friendly interface, high performance graphics, enhanced fringe options, the versatile 2D-plot tool coupled with 3D view, multiplicity of cut planes, drawing windows and models, part statistics, reporting and many others.
The software can effectively handle large models and a wide spectrum of data, including design optimization, composite materials and complex results. The validation of simulation can be greatly assisted by the use of the camera module for the perspective view of the model, the direct matching and overlay of the model view with an image, the video tracking tool and the overlaid display & synchronization of the physical test video with a simulation animation.

BETA CAE Systems S.A.
Kato Scholari, Thessaloniki, GR-57500 Epanomi, Greece
Tel: +30-2392-021420, +30-2311-993300, Fax: +30-2392-021828
Email: ansa@beta-cae.gr, URL: http://www.beta-cae.gr