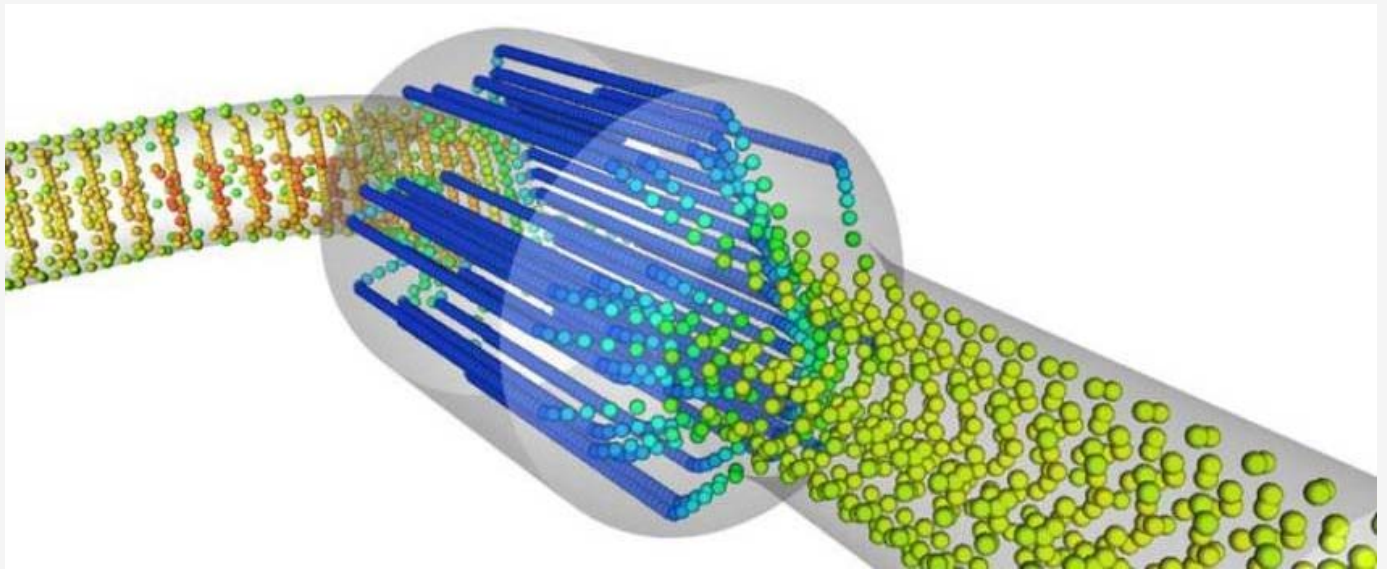


BETA CAE Systems S.A.
announces the release of
ANSA & μ ETA v15.2.0



About this release

BETA CAE Systems S.A. announces the release of v15.2.0. Developed in close cooperation with our customers, v15.2.0 offers a complete package of new solutions for pre- and post- processing and further facilitates the CAE working ways.

Amongst these new tools and solutions the most notable are presented below.

Contents

[Understanding the Software Release Schedule](#)

[Enhancements in ANSA](#)

[Enhancements in \$\mu\$ ETA](#)

[Compatibility and Supported Platforms](#)

[Download](#)

[Documentation](#)

Understanding the Software Release Schedule

The plan

We are committed in delivering improved and enhanced software releases, the soonest possible, in order to meet the requirement of our customers for the continuous improvement of their experience and work. Therefore, we are working in releasing new software versions with code corrections, new software features and enhancements, in regular, frequent intervals.

- A major software version is released every year.

- First point releases, such as v15.1.0, v15.2.0 and so on, with code corrections but also with additional software features and enhancements are released every three months.

- Second point releases, such as v15.2.1, v15.2.2, v15.2.3 mainly with code corrections only upon their parent first point release, are scheduled on a monthly basis.

Each software release is accompanied by a detailed description of the introduced corrections and/or additions so that our customers can decide whether it is critical to implement this release in their environment.

This release

This release of v15.2.0 implements enhancements and code corrections to the v15.1.x branch.



[Enhancements in ANSA](#)

Data Management

As data continue to grow in CAE and the demand for an increased capacity to process CAE data expands, new features and tools to further facilitate Data Management capabilities and reduce time consumption are required. In this version important enhancements have been made in this direction.

“Open In ANSA”: The Product tree Editor which acts as a bridge between the CAD/PDM and the CAE world has been significantly improved offering additional handling tools for reading a model's definition.

The “Open in ANSA” function can handle all supported CAD formats. It assigns automatically the tree structure attributes to the parts and merges them with the hierarchy in order to build the first ANSA assembly.

This process is further enhanced with the usage of ANSA DM. Additionally, the model assembly of big models is significantly speeded-up as the new tool orchestrates the parallel execution of CAD translations in different processes.

Change Representation: A new tool for handling Trim parts has been introduced. Trim parts can be automatically substituted by mass which is distributed to the proper regions of attached parts.

Compare tool: In addition to the PPTX report offered in past versions, it is now possible to export results in XLSX and PDF format which embeds the 3D representation and manipulation (rotate zoom, hide/show parts) of the model.

CAD Data Import

The new CT lib service pack in v15.2.0 allows the translation of Inventor 2014 and SolidWorks 2014 files.

Geometry handling & Meshing

A new tool has been introduced for the creation of ribs consisting of shell elements to save time and enable engineers to further experiment with potential solutions to improve the designs of the products.

In Maritime and Heavy vehicles applications, metal sheets are designed unconnected and are represented by shell meshes. v15.2.0 automatically identifies, extends, and connects the parts, based on the element thickness. This functionality simplifies the modeling stages of complex geometric parts.

Structured solid meshes can now be created by extruding shell meshes or solid facets offering options such as snapping on side walls, or using multiple guiding lines.

Eliminating holes or openings is a fundamental step in exterior Acoustics analyses. A new approach can now be followed through a new tool to fill the gaps on detailed (skin), shell mesh or envelopes. This new capability will save considerable time in Acoustics analyses pre-processing processes.

Enhancing the solutions portfolio for geometry handling, in this version the option to extract faces from curve chains (3d-curves, CONs, edges, 1d elements) has been introduced. It is also now possible to extract a mesh from a cloud of 3D points.

Connections & Assembly

New FE representations and handling tools for Connections add up to the already rich functionality of ANSA for Connections and Assemblies, while the Connections Manager's performance has been significantly improved.

NEW FE-Representations: FE Representations are now available for ROBSCAN connections. The HEXA CONTACT, the RBE3-HEXA-RBE3, and the RBE3-DIAMOND-RBE3. These new FE Representations offer the possibility to create welds of fully customizable shape and to combine several existing functionalities of spotwelds and connection lines.

The PENTA-CONTACT-ON-SOLID FE representation is also now available for SEAMLINE connections. It creates PENTA elements attached to the connected components using a TIED contact. The PENTA-CONTACT-ON-SOLID FE Representation is suitable for modeling T and Y weld joints between solid components.

Additional representations include the CONSTRAINED-INTERPOLATION-SP representation, available for SPOTWELD, GUMDROP and SPOTWELD LINE. This new representation generates single or series of LS-DYNA CONSTRAINED INTERPOLATION SPOTWELD elements.

New options: The new option to “Shrink Width” for ADHESIVE LINE connections allows the local reduction of the width of an ADHESIVE LINE in order to better fit the existing space limitations. Another new option named “Beam Subdivision”, provides several discretization schemes for the bolt shank (Per Row, Uniform, Per Shank Region).

The Check Connections function now runs in parallel. This feature improves significantly the performance of the function in assemblies with a large numbers of connections

For connection lines that are partially realized, the success percentage of connection lines is now reported at the Realization Stas/Succeeded field. This field is automatically filled by the Connection Manager upon realization and it provides a direct way to identify partially realized connection lines

KINETICS Tool

ANSA provides in a single CAE pre-processing environment an integrated Multi-Body analysis tool, the KINETICS. In this version this tool is further augmented.

The Contact based results (normal forces, friction forces, etc.) can now be also calculated and examined within the Results Viewer.

A new tool, named Animated Camera, lets the user play and view animations from an observation view point that changes while the animation progresses

Solutions for CFD analyses

In line with our commitment to continue to enrich our solutions portfolio for all CAE disciplines, additions have also been made in this version. Amongst these, layers are now generated and automatically connected to Link geometry side (link faces), ensuring the exact node matching for periodic boundaries. The export of TAU cdf file format is now also supported.

Solutions for Safety Analysis

Seatbelt: The components creation algorithm has been improved, and now provides options to create constant width, conduct basic quality check of the elements, unfold, and smooth the elements that wrap the part.

Solver interface solutions

Satisfying the need for an even more user friendly and effective modeling, a new wizard has been introduced to assist and simplify the creation of Contacts for Abaqus and ANSYS solver decks.

Additionally:

NASTRAN: A tool for applying poroelastic materials on specified parts/areas has been introduced.

RADIOSS, LS-DYNA: The *PART and *SECTION_ids can now be separated.

Abaqus: *PART, *ASSEMBLY, and *INSTANCE keywords are now read.

Pam-Crash: MULTI STAGE is now supported. Models can be read or created directly in ANSA.

For more details about the new software features, enhancements and corrections please, refer to the Release Notes document.



Enhancements in μ ETA

Stress Linearization

It is common for analysts to perform path analysis of results along a line. Especially for cases where the line of interest lies across the thickness of a component or even a weld root, new functionality is now available to help study these results. The analyst can easily create spreadsheet reports on a per-state or loadcase basis in one step and also produce graph plots of the stress linearization results. This functionality is available in the Parametric Point Paths tool.

Manage Styles

It is always preferable for images of simulations, e.g. within reports, to have a uniform appearance irrespective of the workstation's operating system, the user profile, simulation version and so on.

To meet this demand, a new tool, named Manage Styles, has been introduced. By utilizing the functionality of the tool it is possible to create a set of styles of the current: model drawing style (wire, shade, etc.), scalar and vector fringebar settings (number of colors, values, etc.), visibility of model entities, window settings.

These can then be saved in a .xml file to be shared among different users and/or be selectively reapplied on another μ ETA window or session, even on a different simulation.

CFD – Flow Paths tool

Following the ever increasing functionality added in the CFD capabilities of μ ETA, the Streamlines tool has now been renamed to Flow Paths. Apart from the existing capability to create, display and handle streamlines, the tool can also manage pathlines (flow paths of massless particles in transient simulations) and particle tracks (force-driven paths of mass particles).

NVH - TPA from Forces and FRFs

Performing Transfer Path Analyses (TPA) based on Forces and Frequency Response Functions (FRF) that stem from different files is a procedure which, to be performed manually, requires certain steps. Moreover, manual preparation could not provide options for summing up contribution results to Component and Group of components level.

To remove these constraints and guide the user in performing such analyses a new User Toolbar, named TPA from Forces and FRFs, is introduced offering an intuitive interface with the necessary settings.

Critical Areas

The post-processing of results at critical inner areas of complex structures can be challenging in both manual actions and time.

To help the post processing of results in such cases a new User Toolbar, named Critical Areas, is introduced. The toolbar guides the user to easily set fringe colors to the specific groups of the critical areas while the rest of the model is automatically made transparent, to aid the visibility of those areas. The toolbar also offers functionality to easily create and handle Annotations that point to the critical areas.

Abaqus Interface

Abaqus 6.14 is supported

The keywords ASSEMBLY, INSTANCE and PART are now supported.

VDI criteria can be calculated also on solid elements.

The CZASTATE result for the crushing response of composite materials is now supported.

Geometry and results of Discrete Particle Elements, PD3D, are now supported.

ANSYS Interface

New keywords supported including BEAMxx, LINK180, MP, PIPExx, PLANE182, PSMESH, TARGE170 and more.

Newly supported entities including boundary conditions (F, D, SFE, etc), RBE3 constraints, includes and rotational reaction forces.

Axisymmetric elements can now be displayed as solids.

Now for each contact surface (master/slave) a new property is automatically created.

Newly supported results including Von Mises equivalent results, total mechanical strain, total mechanical thermal strain results and results on beam elements.

EnSight Interface

Block format results are now supported.

FEMZIP Interface

Newly supported FEMZIP-compressed files include FEMZIP-N 1.4.1 Nastran files, FEMZIP-L/P/R 7.84 (LS-Dyna/Pamcrash/Radioss), Pamcrash ERF files and Animator A4 databases.

Plotting of curves directly from FEMZIP-compressed ASCII, Nastran, LS-Dyna, MADYMO, Pamcrash, Pamview and PERMAS files.

FieldView Interface

Files in .fv and .fvb format are now supported.

Fluent Interface

Multiphase results are supported.

IM Interface

FRF results from IM files are now supported.

Nastran Interface

Hyperelastic CTRIA6/CQUAD8 elements are supported

Newly supported results include Poro-Elastic Material and Nastran Embedded Fatigue spotweld/seamweld results.

OpenFOAM Interface

Relative velocity is now supported in MRF results

PAM-CRASH Interface

ERF2 files are now supported

RADIOSS Interface

The keyword PARAMETER and TRANSFORM are now supported.

The MCS (Material Coordinate System) option is now available for stress results.

THESEUS-FE Interface

New keywords and results supported, including manikin FIALA-FE , normal conduction flux, conductivity and resistance results and CONTACTs and VENTILATIONs keywords

U3D Interface

New, export in U3D and U3D pdf format options include isofunctions, vectors created by the Section Forces tool and file compression of the exported U3D and U3D pdf files.

NVH – Modal Response & FRF Assembly

The response DOFs can now be linearly combined.

Multiple Response Frequency Ranges are now supported.

Stresses and other element results available from Nastran SOL103 analyses can now be calculated in the frequency domain.

2D Plot

New filtering options of user-specified Octave filters and convolution filters Moveaverage ,Hanning, Hamming, Blackman, Triangular, Parzen and Welch and .

New curve functions to Simplify, create Convex curves, Percentile function, calculate Level Crossing Rate, Flip, calculate RSS Points.

Isofunctions

Calculation of isofunctions' volume.

Report and Spreadsheet Editor

Right-to-left languages supported.

XLSM files supported.

Occupant Injury Criteria Toolbar

Radioss results are now supported.

Q10 and Q6 child dummies are now supported.

CFD Post Toolbar

Output of Scalar and Vector results in ASCII format that can be input in ANSA.

Map Results Toolbar

Vector Results can now be mapped.

For more details about the new software features, enhancements and corrections please, refer to the Release Notes document.



Compatibility and Supported Platforms

ANSA files saved by all the first and second point releases of a major version are compatible to each other. New major versions can read files saved by previous ones but not vice versa.

The .metadb files saved with µETA version 15.2.0 are compatible and can be opened by earlier versions of µETA.

Support for 32-bit platforms has been discontinued for all operating systems for all platforms.



Download

Where to download from

Customers who are served directly by BETA CAE Systems S.A. may download the new software, examples and documentation from their account on our server. They can access their account through the "user login" link at our web site <http://www.beta-cae.gr>

Contact us if you miss your account details. The [Public] link will give you access to the public downloads area.

Customers who are served by a local business agent should contact the local support channel for software distribution details.

What to download

All files required for the installation of this version reside in the folder named "**BETA_CAE_Systems_v15.2.0**" and are dated as of **November 11th, 2014**. These files should replace any pre-releases or other files downloaded prior to that date.

The distribution of this version of our pre- and post-processing suite is packaged in one, single, unified installation file, that invokes the respective installer and guides the procedure for the installation of the required components.

For the installation of the software on each platform type, the.sh installer file residing in the folder with respective platform name, for Linux and MacOS or the respective .msi installer file for Windows 64bit, have to be downloaded.

In addition to the above, optionally, the µETA Viewer is available to be downloaded for each supported platform.

The tutorials and the example files reside in the folder named "TUTORIALS". This folder includes the complete package of the tutorials and example files, and a package with only the updated ones.

The Abaqus libraries required for the post-processing of Abaqus .odb files are included in the installation package and can be optionally unpacked.

Earlier software releases are also available in the sub-directory called "old" or in a folder named after the product and version number.



Documentation

New ANSA tutorials

NASTRAN: A guide for setting-up NASTRAN SOL600 & SOL400.

CFD: Model comparison and update.

CFD: Catalyst: Case Setup and Morphing for OpenFOAM.

[Updated ANSA tutorials](#)

Basic ANSA

Batch Mesh

Batch Mesh for solid

Assembly of Metal sheets with Spotwelds and seamlines.

