

## New software version release announcement Additional information for the CFD community

May 25, 2009

**ANSA v13.0.1 for CFD**  
**Main new features introduced**  
**Known issues resolved**  
**Download**

### ANSA v13.0.1 for CFD

The new ANSA v13.0.1 has been released, encapsulating numerous new features and corrections for the CFD community users

This version is a follow up of the introductory v13.0 and includes several bug fixes and further improvements. For current v12.1.x and particularly v13.0 users, it is highly recommended to upgrade to this version, in order to take advantage of a robust production version with all the new powerful functionality.

The official software release is comprised by the latest `ansa_v13.0.1` that resides in the server at the time of this announcement.

**This replaces any pre-releases and files downloaded prior to the date : 25-May-2009.**

In order to have the optimum interface and algorithm settings for CFD, remember to start ANSA with the default CFD gui,

e.g. `<installation_path>/ansa_v13.0.1/ansa64.sh -gui cfd`

*Note: if the default fonts of the software are not clearly legible, go to Options>GUI Settings>Fonts and select better ones for the OpenGL, Windows and Menus tabs. Then press Save GUI Settings at the bottom left of the same window, so that the preferred fonts are used next time.*

### For more details...

For details about the new software features and enhancements please, refer to the "**ANSA v13.0.1 Release Notes**" pdf document, that can be downloaded separately and that is also accessible through the *Help>ANSA documentation index* menu bar option within ANSA.

### Main new features introduced

New features that were introduced in ANSA v13.0.1 with respect to v13.0

#### General

- A Star CCM+ I/O was introduced
- Enhanced support of OpenFOAM Deck (several solvers, turbulence models, additional output options and more)
- ISOLATE [Tubes] now works also on FE-mod mesh
- New Selection tool for Elements, Edges, Macros, PID regions etc, appearing at the top right of the main pull down menu when needed).

## MESH

- New function SHELL MESH>COLLAPSE used to collapse needle elements below a user specified length.
- New function SHELL MESH>SUPPRESS New function to smooth out noise and sharp features of a surface mesh.
- Target length for RECONS and RESHAPE in SHELL MESH>PARAM preset as initial local and defeaturing also specified as percentage of local target length, for the CFD gui version of ANSA.
- function VOLUMES>COMPSKIN that can remesh locally FE-mod volume mesh, after modifications (mesh refinement or reconstruction) of its surface mesh.

## Documentation

- New complete ANSA v13.0.1 User's Guide
- Updated CFD Project Practices document with new user defined functions to isolate Macros with specific element type (quad or tria), quick changes of Tolerances and more...
- Updated Tutorials

## New features that were introduced in ANSA v13.0

- A new fully customizable interface
- A default CFD version of the interface (start ANSA with the option: ansa64.sh -gui cfd to enable it!)
- Accelerated graphics and shadow
- New MEASURE Tool
- New functionality to intersect Geom-Fe and Fe-Fe meshes
- New features in A.SPACING and CFD mesh algorithm
- Enhanced RECONS function that respects local length
- Enhanced RESHAPE function to fix all problems on CFD meshed Macros without altering the existing mesh refinement
- New powerful wrapping tool, with variable length capturing curvatures and feature lines
- New flexible Size Boxes to control the mesh size in A.SPACING, CFD and all MESHV. Boxes can have any shape or orientation.
- Improved DETECT function for fully automated Volume definition
- Accelerate Volume meshing with reduced memory footprint
- Improved SHELL MESH>MAP and VOLUMES>MAP algorithms
- REMESH option for all types of Volumes (unstructured and structured)
- New HEXBLOCK volume meshing based on Morphing Boxes
- New LAYERS function, with options for different parameters per PID, zero-thickness walls, auto-connection to neighboring shell mesh and more
- BATCH mesh tool that can accommodate all, Surface, Layers, Volume, so that all meshing is automated and repeatable
- New CFD specific DECKS (CFD++, Fluent, OpenFOAM, StarCD, UH-3D, as well as direct output in CFX5 and SC/Tetra)
- Enhancements in MORPHING and Morphing on Geometry

## Known issues resolved

Some of the known issues resolved in v13.0.1 with respect to v13.0

### General

- Bug and crash fixes in O/I of Fluent, OpenFOAM, CFD++, SC/TETRA.

### TOPO

- Improvements in speed and accuracy of FACEs>INTERS for FE-FE and FE-Geom intersections.

### MESH

- Fixed bug of corrupted perimeters created by RESHAPE or Batch Mesh
- Fixed performance and results of RECONS and RESHAPE functions on Macros and FE-mod mesh
- Fixes in Batch Mesh interface
- Fixed possible crashes in WRAP [Variable length] with defined contact groups
- Improved vector behavior of layers in closed pipes.
- Fixes in Size Boxes

## Download

Our customers may download the new software examples and documentation from their account in our download server. You may access your account through the 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.

### What to download

For the installation of the software on each platform type the following are needed:

1. the tar file with the respective platform name (e.g. sgi, linux etc.), or the respective zip file for Windows and
2. the "common" .tar or .zip file

Previous software releases can be found in the sub-directory called "old".

###

---

For more details contact:

**BETA CAE Systems S.A.**  
Customers Service  
Kato Scholari, Thessaloniki  
GR-57500, Epanomi  
Greece

Tel : +30-2392-021420  
Fax : +30-2392-021828  
email : [ansa@beta-cae.gr](mailto:ansa@beta-cae.gr)  
url : <http://www.beta-cae.gr>