Release Notes for TOSCA Structure 7.0.2 (June 2010)

TOSCA Structure 7.0.2 is a patch release with small improvements and fixes. TOSCA ANSA Environment is additionally updated to version 13.0.5.

TOSCA ANSA environment

- Problem with Abaqus *STEP definition for Validation output is solved.
- Importing an existing parameter file opens the complete Task Workflow (including Output item, Run, Postprocessing, and Smoothing)
- *.ansa_tosca is added to the default file extension when opening databases
- Path length in group based smoothing extended to 1024 characters
- Screen output window remains open after optimization process has finished

TOSCA Structure.Topology

- Reference values in frequency response are fixed

TOSCA Structure.Shape

- Problem with local coordinate systems in manufacturing constraints is solved
- COMBINE can now be used as OPER in DRESP for TOSCA Structure.shape

Abaqus interface

- Abaqus 6.9.3 is now supported.

Ansys interface

- Support for large .emat files (>4Gb) added

NX Nastran interface

- Problem with PARAM,DBALL,SCRATCH and NX Nastran contacts solved

Release Notes for TOSCA Structure 7.0.1

TOSCA Structure 7.0.1 is a patch release with many small improvements and fixes. One of the main changes is a new version of TOSCA ANSA Environment (version 13.0.3).

TOSCA Structure General

- Windows 7 is supported.

TOSCA ANSA environment

The TOSCA ANSA environment is based on the TOSCA ANSA environment release 13.0.3. which has a range of new user friendly functionality:

- Import of existing parameterfiles and automatic display in the Task Manager
- Support of NX.Nastran and NX.Nastran Desktop
- Optimization Job Monitor during interactive RUN OPTIMIZATION
  - View Convergence Plot
  - Generate and View Postprocessing information in TOSCA Structure.view
  - View TOSCA.OUT
  - Abort the TOSCA Structure Job.
- "VIEW" Item for TOSCA Structure.view items and TOSCA.OUT Files.
- Once generated Postprocessing files can directly be viewed just using the VIEW in the context menu of the POSTPROCESSING items without the necessity to update the complete generation of postprocessing files.

- Text Edit of Output Items
  The output item may be edited so the user can check the content of the TOSCA Structure parameterfile in ASCII format. Additional commands and comments can be added manually before submitting the TOSCA Structure job.
- Multiple Smooth Instances
  Multiple Smooth instances may be added to the optimization task to define specific smoothing tasks for validation, CAD-Transfer or visualization that have different requirements e.g. on the mesh size.
- Multiple Cutting Splines per Smoothing step
  Definition of multiple normal vectors for the generation of the cutting splines. All splines for all directions are automatically exported to a single IGS File.
- Group Based Smoothing available
- Solver Groups handling
  The groups that are available in the finite element input deck are imported and displayed in the GROUPS section of the Task Manager.
- Improved Highlightning
- Mesh Smooth Item added for Standard Shape Optimization Tasks
- Reading of existing parameter files

TOSCA Structure.Smooth

- Performance enhancements for large models under Linux
- Calculation of slice intersection points, modified slice data reduction method

TOSCA Structure.Topology

- Improvements of manufacturing constraints for sensitivity based topology optimizations

TOSCA Structure.Shape

- Volume constraint in shape has been modified to reach target volume in design cycle 1. Problems with volume constraint has been solved
- New mesh_smooth algorithm has been implemented. To be used when the normal MESH_SMOOTH fails. To activate use the MESH_SMOOTH item STRATEGY = LOCAL_GRADIENT.
  Although the new algorithm is effective for some problems it has some deficits when used with manufacturing constraints (DVCON_SHAPE). Therefore, the old mesh smooth algorithm is still the default because it is more robust.
- Inner nodes are allowed in design area to ease problem definition
- Problems with GROUP_AUTO_DEF has been solved

Abaqus interface

- Abaqus 6.9.2 is now supported.
- Some types of prescribed displacements, velocities and accelerations being different from zero are now
- supported in frequency response. Prescribed displacements, velocities and accelerations for ABAQUS are
- supported in frequency response using the commands *BOUNDARY including the following: -
- TYPE=DISPLACEMENT - TYPE=VELOCITY - TYPE=ACCELERATION

Ansys interface

- Some types of prescribed displacements, velocities and accelerations being different from zero are now supported in frequency response. Prescribed displacements and thereby indirectly velocities and accelerations for ANSYS are supported in frequency response using the command D for defining degree-of-freedom constraints at nodes.

Msc.Marc interface

- Contact definitions are now supported for sensitivity based topology optimizations including displacement constraints

NX Nastran interface

- NX Nastran 7.0 is supported

NX Nastran desktop interface

- NX Nastran Desktop 7.0 is supported

Permas interface

- Contact definitions are now supported for sensitivity based topology optimizations including displacement constraints

Known limitations in TOSCA Structure 7.0.1

TOSCA Structure.Control

- Special characters (ö, ü, ä, ß, é, etc.) may not be used as file/job names for neither parameter file nor FEM_INPUT files names.

TOSCA Structure.Shape

- Combinations of manufacturing constraints may lead to not manufactorable designs or mesh quality problems.
- CHECK_MAX_MEM (max member size) has some problems especially by 3D models. One known effect is a very distorted surface mesh.

Msc.Nastran, NX Nastran and NX Nastran Desktop

- SPCD is not supported for frequency response.

Release Notes for TOSCA Structure 7.0

TOSCA ANSA™ environment – a new powerful graphical user interface for the leading integrated optimization solution TOSCA Structure

- Interactive GUI for easy optimization preprocessing and postprocessing
- Fully automated generation of validation models (automatic remeshing) for smoothed topology optimized designs from TOSCA Structure.topology

TOSCA ANSA™ environment is a part of the standard installation package but requires a separate license. Regarding licensing, please contact FE-DESIGN GmbH info@fe-design.de

TOSCA Structure general

- TOSCA Structure 7.0 is a very stable version mainly based on TOSCA Structure 6.2.1 with minor changes to enhance the interface for the TOSCA ANSA™ environment and ANSA™
- TOSCA Structure 7.0 requires a new license. Your existing Tosca Structure 6.0 license will not work with Release 7.0. Please contact FE-DESIGN GmbH info@fe-design.de or your local distributor to get an updated license file.
- The documentation for TOSCA Structure has been revised due to the new optimization process applied in the TOSCA ANSA™ environment
- An easy introduction to the TOSCA Structure and the TOSCA ANSA™ environment is now available using a start tutorial, see <tosca_installation_directory>\docu\TOSCA_Structure_v700.pdf

TOSCA Structure Topo

- Manufacturing constraints have been improved. Especially, manufacturing constraints including casting directions in the sensitivity based topology optimization have been enhanced.

TOSCA Structure GUI

- Several bugs have been corrected

TOSCA.Smooth

- Adjustments have been implemented for improving the automated remeshing process of the TOSCA Structure.topology results

ABAQUS Interface

- ABAQUS 6.9.1 is supported

ANSYS Interface

- Ansys v12.0 is supported

NX Nastran

- NX Nastran 6.1 is supported

NX Nastran Desktop

- NX Nastran Desktop 6.1 is supported
- Now supports multiple input decks

TOSCA control and configuration

- The configuration variables:

${__FE_MODEL__} Holds the current FE-model name without file extension
${__FE_FILE__} Holds the current FE-model name with file extension
${__LIFE_MODEL__} Holds the current LIFE-model name without file extension
${__LIFE_FILE__} Holds the current LIFE-model name with file extension
May now be used when calling scripts.

Know limitations in TOSCA Structure 7.0

TOSCA Structure.post

- It is currently not possible to generate .rst files (Ansys) with TOSCA Structure.post

Marc Interface

- The New-Styled Tables (now Default in Marc 2007) are not allowed in Marc input decks for TOSCA Structure. Please switch to the non-table format when exporting Marc input files which will be applied in TOSCA Structure. (This parameter can be changed in the job settings of Mentat)

Permas Interface

- Contact definitions in sensitivity based optimizations are currently not supported

Ansys Interface

- MESH200 elements are not allowed in the input deck

Discontinued platforms

- The platform IBM AIX 5.3 Power will be deprecated for future versions of TOSCA Structure due to the descending interest regarding the present platform. IBM AIX 5.3 Power for TOSCA Structure 7.0 will still be available on request. Please contact FE-DESIGN GmbH for further questions regarding this platform.

Deprecations

Following TOSCA Structure.post-formats will be cancelled for future versions of TOSCA Structure.post:

- op2
- rst
- odb
- bof
- unv

The vtf-format will remain the main visualization media for viewing TOSCA Structure results. Please contact FE-DESIGN GmbH if one of the above TOSCA Structure.post outputs is an embedded part of your optimization workflow.

Contact

In case you have further questions, please do not hesitate to contact our support team:

TOSCA Structure Support
FE-DESIGN GmbH
Haid-und-Neu-Straße 7
76131 Karlsruhe
German

Tel.: +49-(0) 721-96467-250
Fax: +49-(0) 721-96467-290
Mail: support@fe-design.de
Web : www.fe-design.com