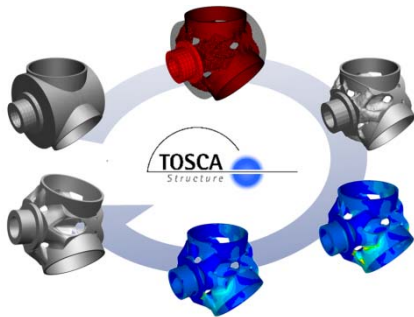


TOSCA Structure 7.1

TOSCA Structure is a modular system for non-parametric structural optimization that enables topology, shape and bead optimization in combination with standard finite element solvers. This allows for a seamless integration in existing CAE environments and guarantees easily interpretable and reliable results. The setup is very simple as no model parametrization is necessary and existing solver input files can directly be used for optimization.



Optimization with TOSCA Structure

TOSCA Structure supports the complete optimization workflow including result transfer to CAD systems and automatic model verification. A closed development process can be achieved from the first concept to the optimized geometry in the CAD system.

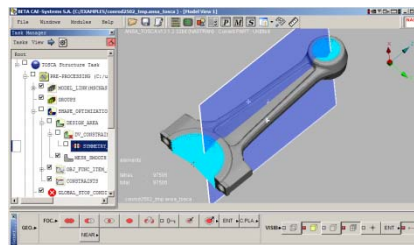
General capabilities

- Solver interfaces: ABAQUS, ANSYS, Marc*, MSC.Nastran, NX Nastran, NX.NASTRAN Desktop and PERMAS* (*frozen with TOSCA Structure 7.0.1)
- Stable and fast optimization algorithms
- Direct use of existing models
- Efficient handling of very large models
- Support of unlimited number of load cases and boundary conditions
- Consideration of multiple FE analyses for response combination
- Optimization in combination with non-linear analysis (contact, non-linear material, large deformation)
- Pre- and postprocessing of optimization jobs in TOSCA ANSA environment and TOSCA Structure.gui

- Visualization of optimization results using TOSCA Structure.view
- Combination of postprocessing data to reports using TOSCA Structure.report
- Result preparation for CAD transfer using TOSCA Structure.smooth

TOSCA ANSA environment

The optimization setup can be carried out in TOSCA ANSA environment. Based upon the executable finite element input deck of the initial or design space model, all optimization settings are defined interactively.



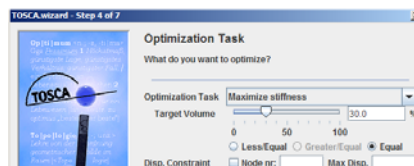
Setup with TOSCA ANSA environment

TOSCA ANSA environment supports the complete optimization workflow:

- Setup of the optimization task
- Check model consistency
- Run optimization
- Postprocessing and Visualization
- Generation of smoothed results
- Automatic model verification

TOSCA Structure.gui

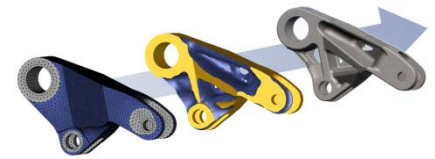
The classical user interface TOSCA Structure.gui also supports the complete optimization workflow. An easy-to-use wizard allows the setup of standard optimization tasks in only a couple of mouse clicks.



Setup with TOSCA Structure.gui

TOSCA Structure.topology

Topology optimization determines an optimum design proposal starting with a given maximum design domain.



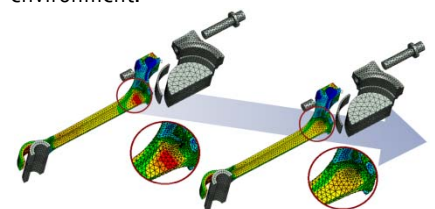
Topology optimization of a lever
(Images courtesy of JCB)

All loads and boundary conditions from the finite element model are taken into consideration during the automatic modification procedure.

- Free choice of objective function terms and constraints of the following design responses: volume, mass, compliance, displacements, rotations, reaction forces, reaction moments, internal forces, natural frequencies, center of gravity, moments of inertia, frequency response analysis results (amplitudes, phases, velocities and accelerations), acoustic measures (surface velocities, sound pressure)
- Demolding constraints for casting and forging components. Consideration of stamping conditions and tightness
- Various symmetry constraints (plane, rotational, cyclic) and link conditions
- Minimum and maximum membersize filters for wall thickness constraints
- Fully automated generation and analysis of verification models

TOSCA Structure.shape

In shape optimization, the component's surface is modified to reduce local stress or damage peaks. For the setup of a shape optimization problem in TOSCA Structure no model parametrization is necessary. The design area is simply defined via group definitions in the finite element model or directly in TOSCA ANSA environment.



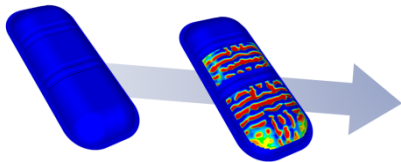
Shape optimization of a connection rod
(Images courtesy of Ford Werke AG)

- Minimization of combinations of equivalent stress values (various stress hypothesis available).

- Maximization of selected natural frequencies
- Specification of a volume constraint
- Mesh-independent manufacturing constraints for casting, forging, stamping, extrusion and drilling
- Minimum and maximum member size
- Various symmetry constraints: plane symmetry, rotational symmetry
- Penetration checks to neighbouring parts
- Mesh smoothing in each design cycle to keep high quality meshes during optimization

TOSCA Structure.bead

Bead optimization is used to determine the optimum location and orientation of bead stiffeners for sheet metal components. With TOSCA Structure.bead, the static stiffness and the vibration behaviour can be improved.



Bead optimization of a muffler
(Images courtesy of Tenneco)

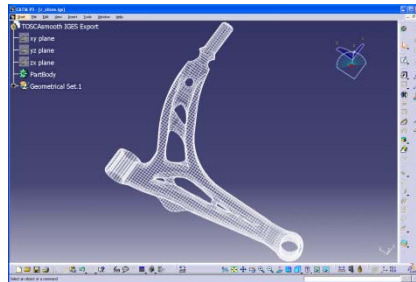
- Definition of objective function terms and constraints using the following response types: compliance, natural frequencies, displacements, results from frequency response analysis (amplitudes, phases, velocities and accelerations), acoustic responses (surface velocities)
- Symmetry constraints
- Penetration checks to neighbouring parts
- Definition of maximum bead height
- Mesh-independent approach
- Easy interpretation of the results

TOSCA Structure.smooth

The transformation, smoothing and data-reduction of optimization results, the preparation of validation models and the export to CAD-systems is realized with TOSCA Structure.smooth.

- Calculation of smoothed isosurfaces of the material distribution after topology optimization depending on isovalue or target volume
- Generation of patched surface geometries for shape and bead optimization

- Visualization in TOSCA Structure.view
- Data reduction for CAD transfer
- Calculation of cutting splines and export as IGES files
- Export of patched surfaces in CAD compatible formats (STL, IGES)
- Export as FE surface mesh for manual remeshing and analysis or automatic reconstruct and verification run



Cutting splines in CATIA V5.
(Image courtesy of Otto Fuchs KG)

TOSCA Structure.advanced

For high level optimization applications TOSCA Structure offers additional modules that allow to use advanced functionalities:

LIFE

- Shape optimization based on results of fatigue simulation
- Use of FEMFAT, FE-Safe, Falancs, nCode DesignLife, Femsite or user specific fatigue code possible

NONLINEAR

- Topology and shape optimization of contact models
- Topology and shape optimization using non-linear material behaviour
- Topology and shape optimization based upon geometric non-linear analysis

ACOUSTICS

- Support of acoustic measures (surface velocities, sound pressure)
- Support of results from frequency response analysis (amplitudes, phases, velocities, accelerations)

TOSCA Structure.report

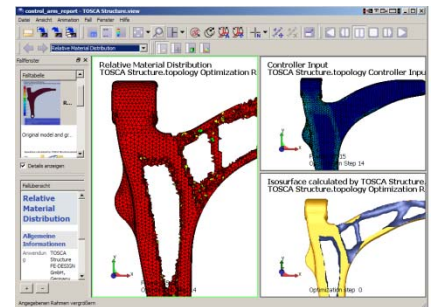
TOSCA Structure.report captures TOSCA Structure optimization results for easy and reproducible postprocessing.

- Collection of screenshots, fringe plots, animation sequences and x-y-plots of constraint and objective function values in vtfx archives

TOSCA Structure.view

TOSCA Structure.view is a free, high-performance 3D viewer for TOSCA Structure data in vtfx format.

- Viewer for platform independent, binary and compact vtfx archives containing 3D animations and x-y-plots of the optimization history
- Available as stand-alone application on Windows and Linux
- Available as plug-in for Office applications or Internet Explorer on Windows



TOSCA Structure.view

Training

For an efficient use of TOSCA Structure we recommend a consulting project or an initial training.

Current training dates are listed on our website www.fe-design.com.

Contact

Get in touch with us

- if you want to evaluate TOSCA Structure
- if you need more information
- if you are looking for a competent and flexible engineering service provider, who can solve your analysis and optimization problems.

FE-DESIGN GmbH
Haid-und-Neu-Strasse 7
76131 Karlsruhe
Germany

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

TOSCA is a registered trademark of FE-DESIGN GmbH. All other trademarks are the property of their respective owners.

© 2011 FE-DESIGN GmbH