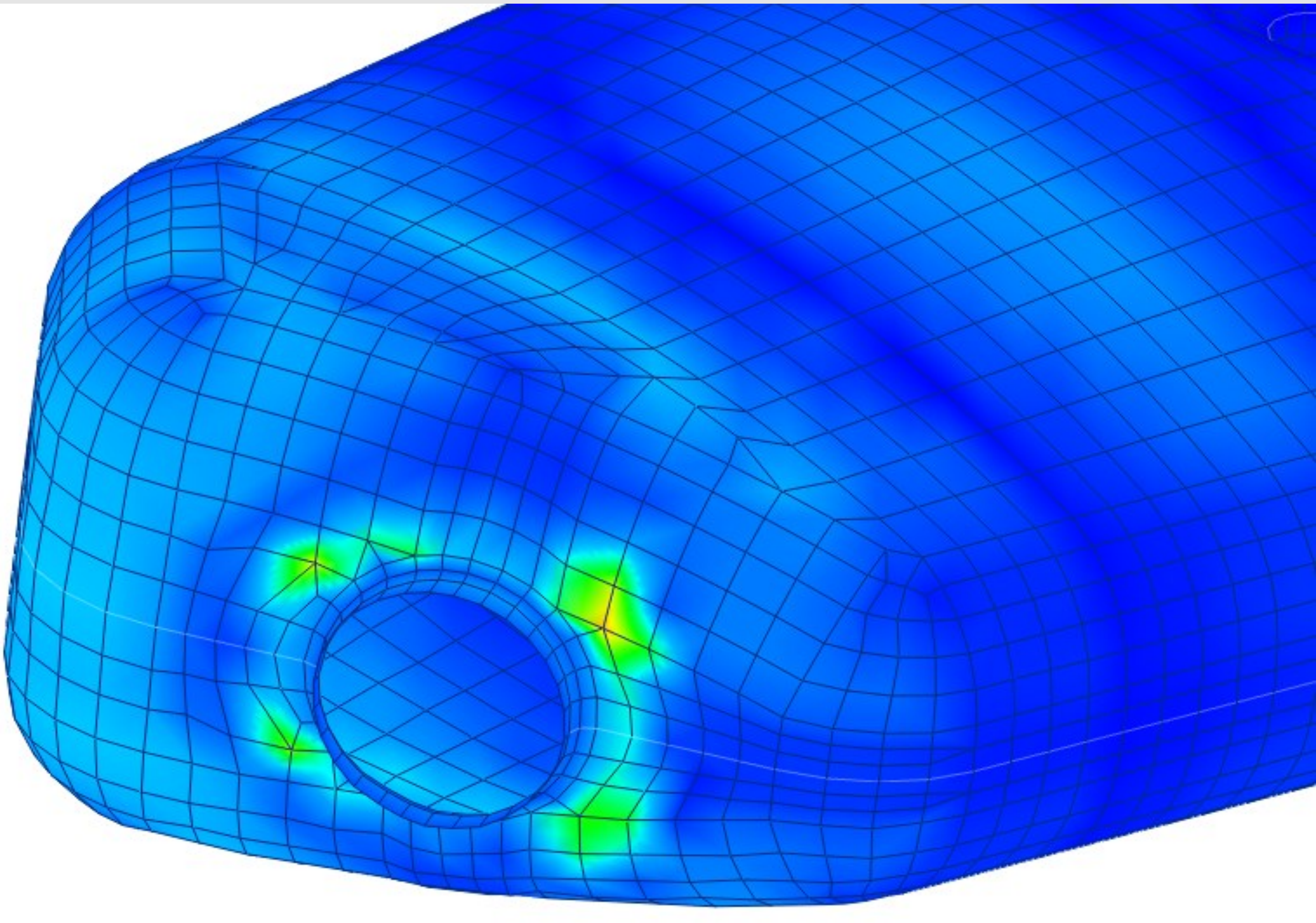
FEMtools Product Overview

Essential Tools for Finite Element Based Simulation, Validation, Updating and Optimization



FEMtools simulate > validate > update > optimize

Dynamic Design Solutions

Test-Analysis Integration

Designing products today offers many challenges. They must be stronger, lighter, safer, less expensive, and new materials are being used. Products have to satisfy a widening range of design criteria, including environmental impacts.

To keep development time and cost competitive, industry relies on computerized simulation tools. Finite element analysis (FEA) is a powerful technique to simulate the behavior of a product under various types of loadings.

The FEA method has matured over the past three decades to a point where design, meshing, analysis and postprocessing are becoming highly integrated and automated. This predictive approach relies on the quality of the simulation model, the software to analyze it and the engineering judgment of the analyst interpreting the results.

In order to keep up with quality requirements, simulation models and procedures must be validated. A good way to do this is testing. Experimental methods to support product design and analysis are based on prototype measurements under laboratory conditions or in real-life situations. They are effective to learn about the product, and the environmental conditions.

In a competitive world, however, a trial-and-error design and analysis approach involving a series of prototypes is too time-consuming and expensive. It is therefore necessary to reduce the number of iterations on prototypes. This can be achieved by deriving more information on prototype testing and by shifting prototyping to later in the design process.

Integrating test and simulation enables synergistic processes from which the entire engineering team can benefit. Some examples:

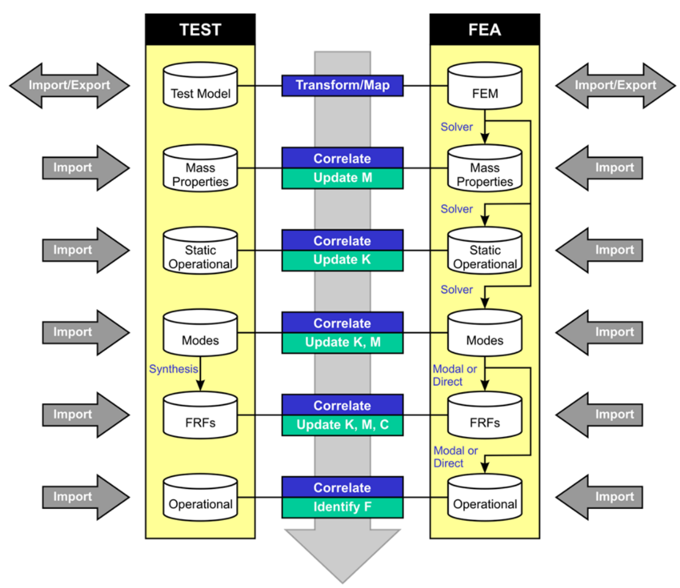
- FEA results can be used to optimize the experimental setup (pretest analysis)
- Mixed numerical-analytical methods are used to quickly and reliably identify otherwise only approximately known structural properties (e.g. joint stiffness), material properties, and loading (non-destructive, indirect, testing).
- Test results are used as reference data to validate and refine a finite element model (error localization, correlation analysis, model updating). Unknown or badly known physical properties can be identified and uncertainties in finite element models better assessed.
- Hybrid models that contain partially FE models and partially test models can be developed to build more complete models that include all essential components while maintaining a good balance between model size and performance.

Model validation should be part of every modern engineering analysis quality assurance procedure (ref. ISO 9001). Many parameters in a finite element analysis are uncertain.

FE model validation is the demonstration that a simulation model is fit for purpose and results can be used with less uncertainty.

By better exploiting the combined test and analysis data, engineering staff and management will gain more confidence in the simulations they increasingly need to rely on for applications related to NVH, and involving design optimization, acoustics, and fatigue analysis. They will also gain expertise for future modeling, build up reusable knowledge and more rapidly acquire essential skills like engineering judgment.

Most obstacles to combining test with analysis have been removed in recent years. With FEMtools, Dynamic Design Solutions offers a software suite that satisfies the needs of the engineering community for a dedicated toolbox for integrating test-analysis, with a focus on model validation. It contains all the essential components and is open, customizable, easy to use and available on all popular hardware platforms.



Configurations and Add-ons Overview

FEMtools is a multi-functional, cross-platform and solver-independent family of CAE software programs providing advanced analysis and scripting solutions in the following areas:

- Test-analysis integration
- Simulation and structural optimization
- Process integration and automation
- Engineering database management

There are six FEMtools configurations, each targeting different types of applications:

- **FEMtools Framework** — An Interactive desktop and scripting environment for engineering analysis and CAE process automation.
- **FEMtools Dynamics** — Advanced finite element solutions for simulating dynamic response and structural modifications.
- **FEMtools Pretest and Correlation** — A solution for modal pretest analysis and model validation using test-analysis correlation.
- **FEMtools Model Updating** — An integrated solution for structural dynamics simulation, model validation and updating.
- **FEMtools Optimization** — An integrated solution for structural design optimization.
- **FEMtools Model Updating and Optimization** — A complete solution combining FEMtools Model Updating and FEMtools Optimization.

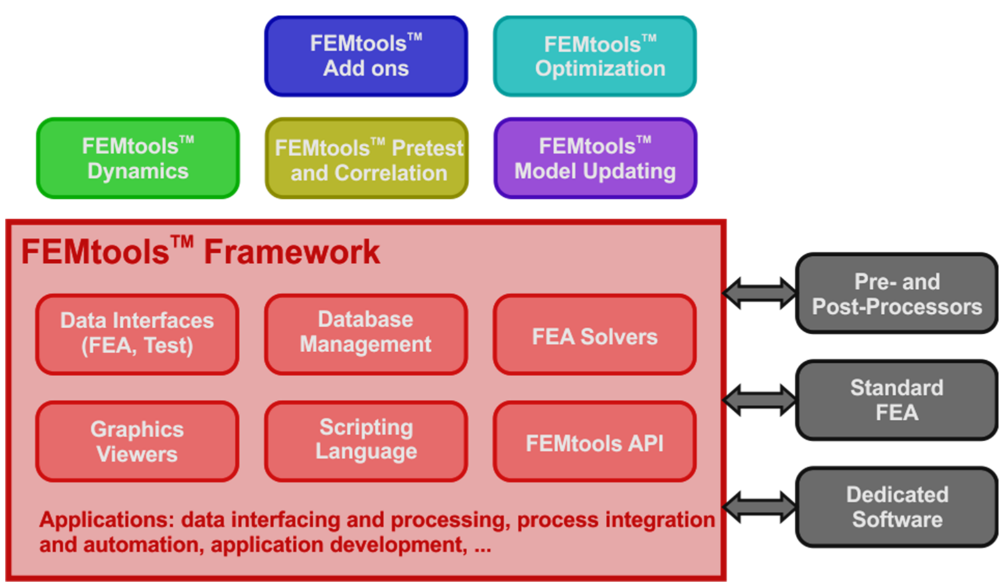
The program is available on all popular engineering computing platforms and integrates in an existing environment.

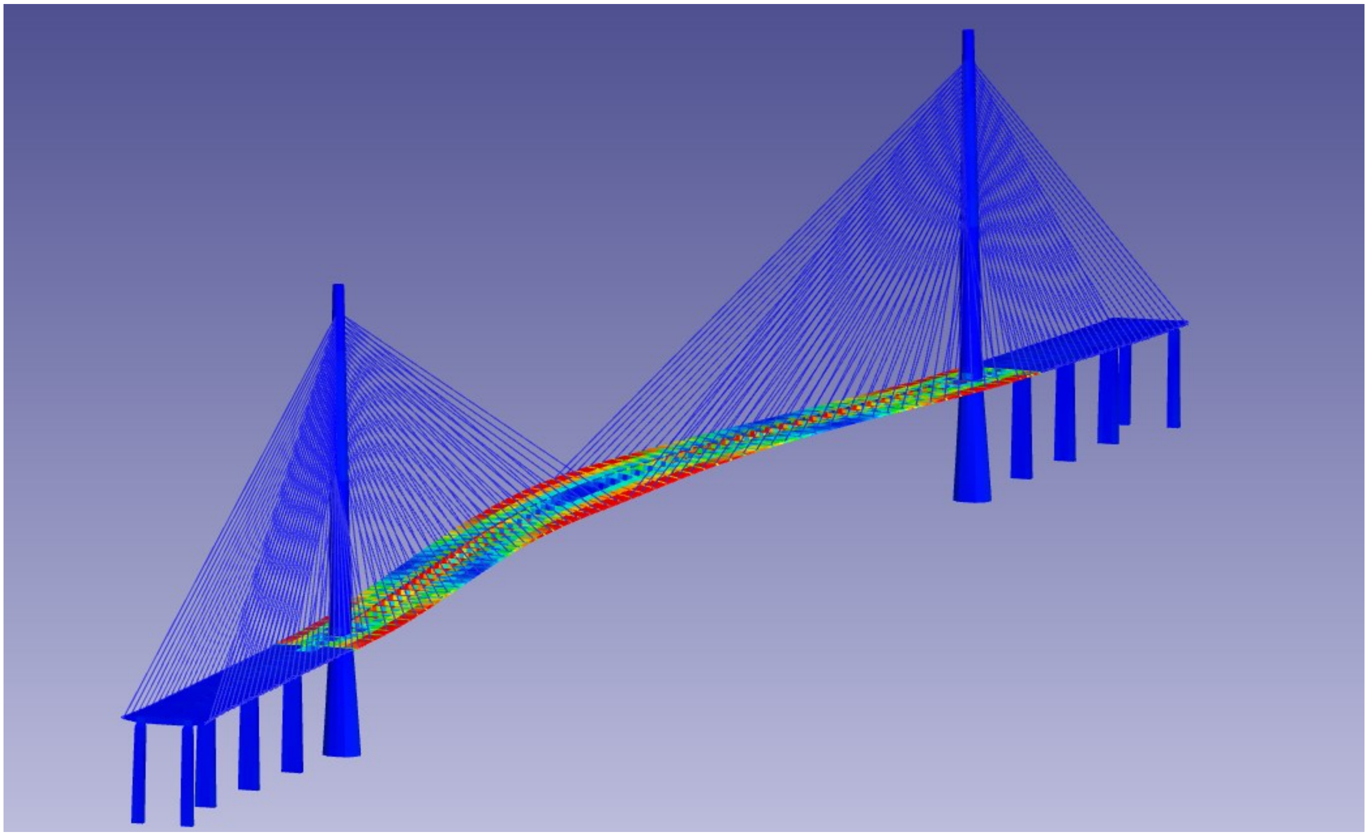
Each FEMtools configuration can be complemen-ted with the following add-ons for integration with standard commercial FEA programs or for additional specialized functionality:

- **ABAQUS Interface and Driver** – Interfacing and piloting the ABAQUS general purpose finite element program.
- **ANSYS Interface and Driver** – Interfacing and piloting the ANSYS general purpose finite element program.
- **LS-DYNA Interface and Driver** – Interfacing and piloting the LS-DYNA general purpose finite element program.
- **NASTRAN Interface and Driver** – Interfacing and piloting the NASTRAN general purpose finite element program.
- **SAP2000 Interface and Driver** – Interfacing and piloting the SAP2000 finite element program for civil engineering simulations.
- **FEMtools MPE** – Powerful modal parameter extraction using local and global methods.
- **FEMtools RBPE** – Rigid body properties extraction from FRFs.
- **FEMtools DAQ** – Data acquisition using National Instruments frontends.

For test data, FEMtools can read and write Test Universal Files or use custom scripts for additional interfacing solutions.

FEMtools is designed to handle all sizes of finite element models for a wide range of structural analysis applications in different industries (automotive, aerospace, marine, civil engineering, power generation, machinery,…).





FEMtools Framework

An Interactive Desktop and Scripting Environment for Engineering Analysis and CAE Process Automation

What is FEMtools Framework?

FEMtools Framework is a multi-functional interactive environment for advanced engineering application development, integration, and automation. The framework includes data interfaces, database management utilities, mesh generation and manipulation tools, parameter and response management, state-of-the-art data visualization, plus a full featured scripting language and API function library.

Optionally, FEMtools Framework includes a standard finite element library and solvers for linear static and normal modes analysis. Alternatively, standard external solvers like NASTRAN, ANSYS, ABAQUS, LS-DYNA or in-house solvers can easily be integrated and piloted as part of larger analysis processes.

This open and flexible CAE application development platform is used by analysts to integrate their tools, and create vertical applications that meet the specific requirements of an industry. A unique capability of FEMtools Framework is the integration of data resulting from experimental static or dynamic testing.

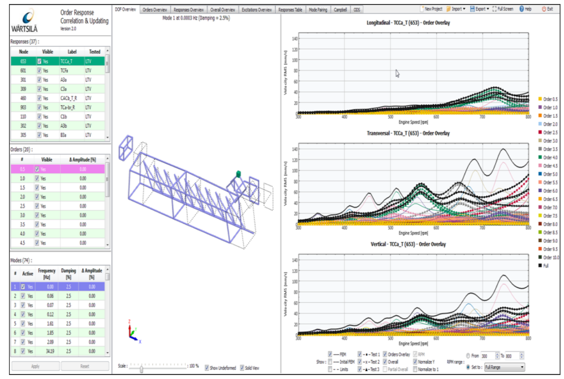
FEMtools Framework provides the foundation for all FEMtools application modules, data interfaces and add-on tools.

Furthermore, a growing number of development partners are working on integrating their existing tools or developing entirely new tools that exploit the framework infrastructure and re-usable components. This is a more efficient approach than developing new applications from the ground-up.

As a stand-alone tool, FEMtools Framework can be used as a utility software for

- Data management, translation and transformation
- Pre- and postprocessing of FEA and test data.
- CAE process integration and automation.
- Development of vertical applications.

FEMtools Framework configuration adds the following modules to the framework:

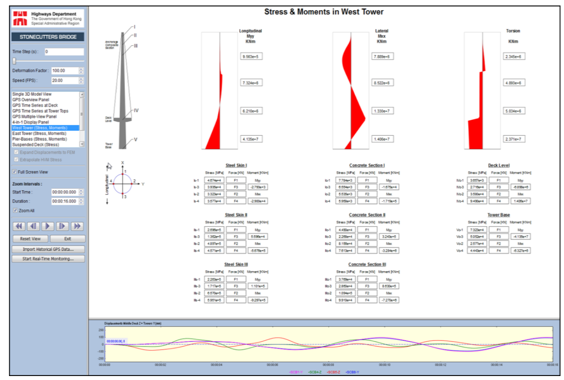


Database Management

Utilities for interactive definition, editing and transforming data. Includes alignment of finite element and test models, expansion and reduction of databases. Data exchange with external programs is provided by optional add-on modules.

Mesh Generation and Morphing

A function library is available to generate surface meshes and volume meshes. Unstructured surface meshes can be generated for generic surface patches that are defined using simple geometry description (vertex, curve, surface). Volume meshes are obtained from 2D meshes with operations like extrusion and revolving.



This FEMtools API function library complements the lattice-based mesh deformation ('morphing') and provides the functionality to mesh very complex geometries. It can be used for applications like parametric meshing for shape optimization, meshing design spaces for topology optimization or to implement mesh coarsening and refinement tools

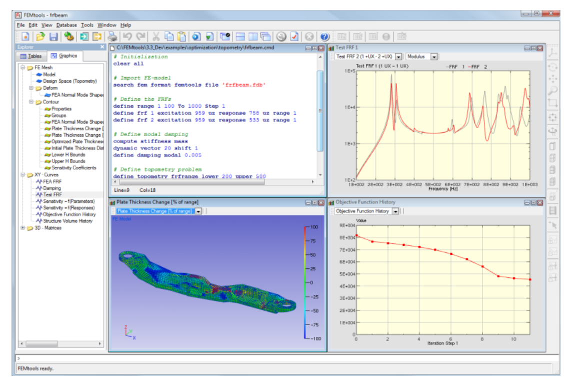
Mesh Quality Analysis

Mesh quality analysis tools verify the quality of imported or generated finite element meshes. Quality metrics like aspect ratio, convexity, taper, warp angle and others can be computed and visualized with color-coded mesh graphics.

Finite Element Library and Solvers

An internal library of finite elements and materials is included for stand-alone static and normal modes analysis.

Parameter and Response Management



Commands for defining parameters and responses are included. Parameters are physical element properties that can be used as variables for sensitivity analysis, design optimization or model updating applications. The definition can be based on existing property cards or new properties will be generated of selected elements or sets of elements.

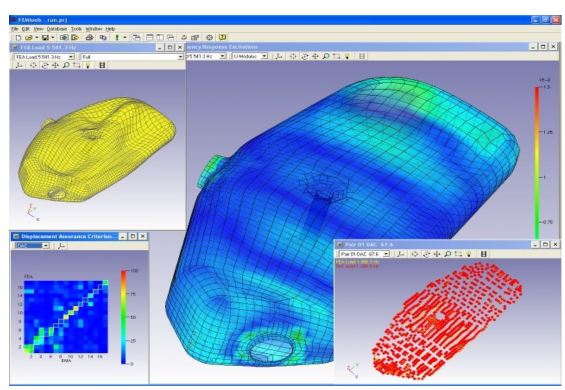
Responses can be mass, displacements, strain, stress, modal parameters. MAC, FRFs or ODS. They serve as targets for sensitivity analysis, model updating and design optimization.

User Interface

FEMtools is operated via a customizable menu interface, the FEMtools command language and FEMtools scripting.

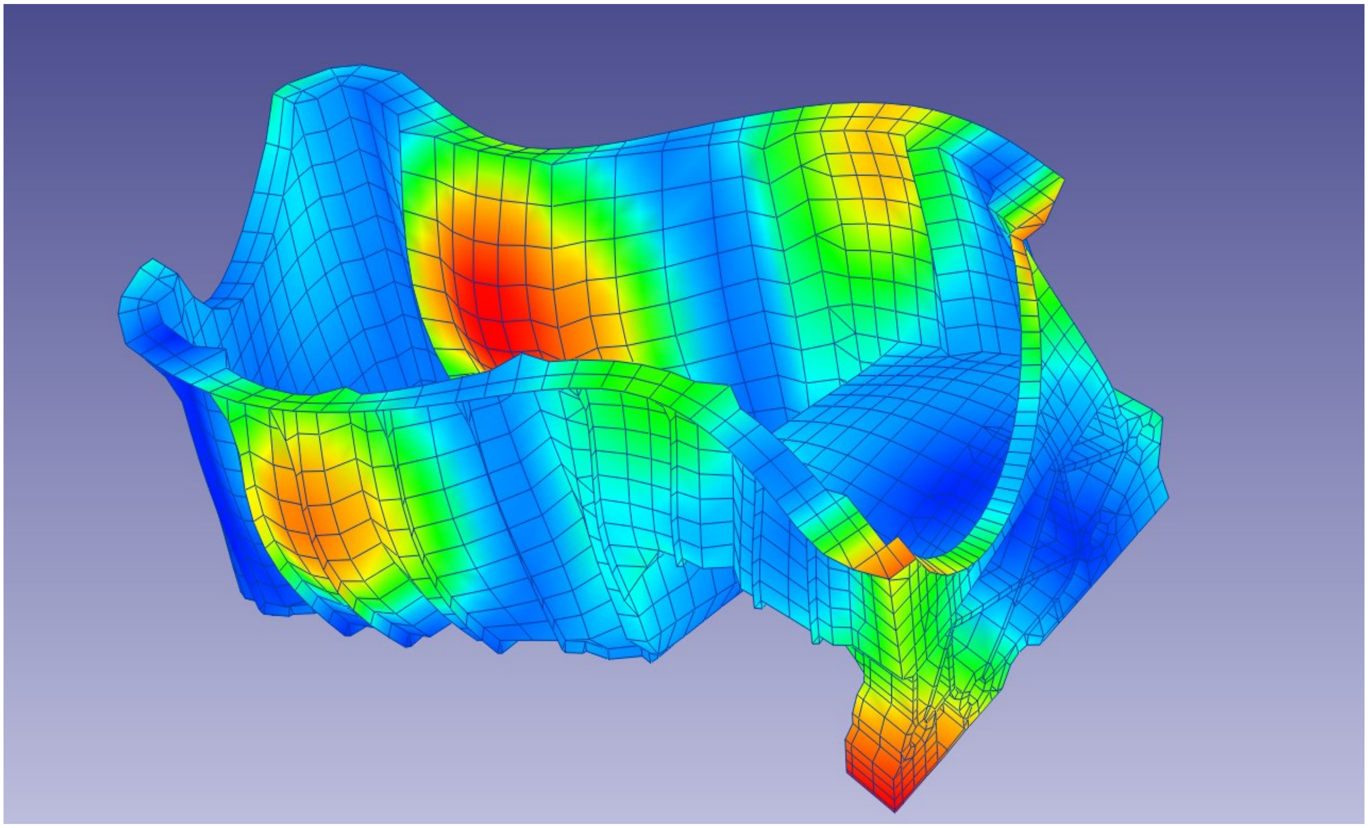
Graphics Viewers

Highly interactive tools for dynamic 3D visualization of finite element and test models, animated shapes and all FEMtools analysis results. Support for high resolution monitors (4K).



FEMtools Script Language and API

Procedures can be automated or customized using an integrated scripting language and API for database access, process control, and user interface programming.



FEMtools Dynamics

Advanced Finite Element Solutions for Simulating Dynamic Response and Structural Modifications

What is FEMtools Dynamics?

FEMtools Dynamics is a FEMtools configuration that brings together a number of analysis tools that are commonly used in advanced structural dynamics simulations.

Computation of Frequency Response Functions (FRF) is provided in a way that these FRFs can be directly correlated to equivalent test FRFs. Frequency range and resolution, input and output locations and damping can be defined similar to the actual vibration tests that are performed on a structure.

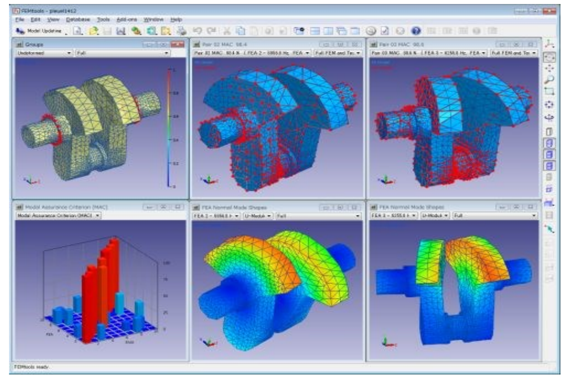
Different computation methods are implemented, i.e. modal or direct solvers, using unit single point forces simulating hammer or shaker excitation, or realistic operational loads. Advanced methods like residual vectors, dynamic compensation, or Padé approximants are available.

Another key technology is dynamic substructuring. This includes methods like Craig-Bampton superelement analysis, modal-based assembly (MBA) and FRF-based assembly (FBA). These methods are used to accelerate analysis times but also support the bottom-up approach to the validation of assembled structures.

Structural Dynamics Modification (SDM) is a design-oriented tool to rapidly simulate effects of structural changes on the dynamics of structures. A library of lumped, 1D, 2D or 3D modification elements is available to model most commonly encountered structural modifications. All analysis is done in modal space.

Time Domain Simulation (TDS) provides a set of tools to compute the transient response of structures in a computational efficient way. The FEMtools TDS solver first derives a state-space model from the normal modes of the structure and then uses this model to compute the time series of the responses. It is possible to completely simulate a modal test by combining TDS with the FEMtools MPE add-on tool.

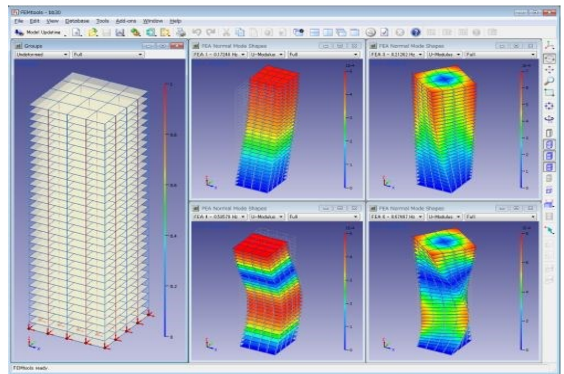
The FEMtools Dynamics configuration adds the following modules to the framework:



Frequency Response Analysis

- Compute FRFs comparable to experimentally obtained FRFs.
- Direct FRF computation with support for multi-reference enforced displacements.
- Padé approximant method with automatic convergence control.
- Modal FRF synthesis from FEA or test modes.
- Residual vectors (inertia relief, viscous damping and applied loads)
- Dynamic compensation method for improved modal FRF.
- Support for modal, proportional viscous and structural damping, viscous damper elements, and material damping).
- Support for local coordinate systems.

Harmonic Response Analysis



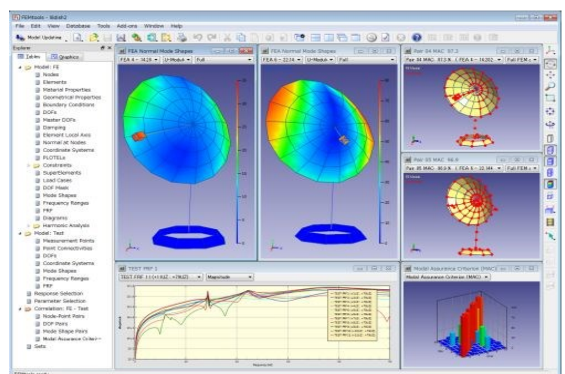
- Operational shapes analysis using modal and direct solvers.
- Displacement, velocity or acceleration responses functions.
- Enforced response analysis.

Superelement Analysis

- Integrated Craig-Bampton matrix reduction.
- Easy definition of superelements using sets of elements or nodes.
- Support for assemblies without residual model.
- Automatic generation of master DOFs at test locations.
- Support of dependent DOFs in DOF relations as master DOFs of a Superelement.

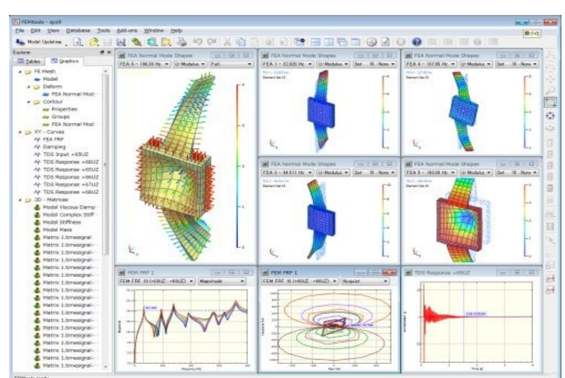
Modal-Based Assembly

- Modification of a finite element model or a test model using individual modification elements (finite elements), using tuned absorbers or modal substructures.
- Computing modified mode shapes and resonance frequencies using modal parameters from test or finite element analysis.
- Correlation analysis between unmodified and modified models.
- Variational analysis of all physical properties of the modification elements using the fast modal solver.
- Fast modal solver for model updating and optimization.



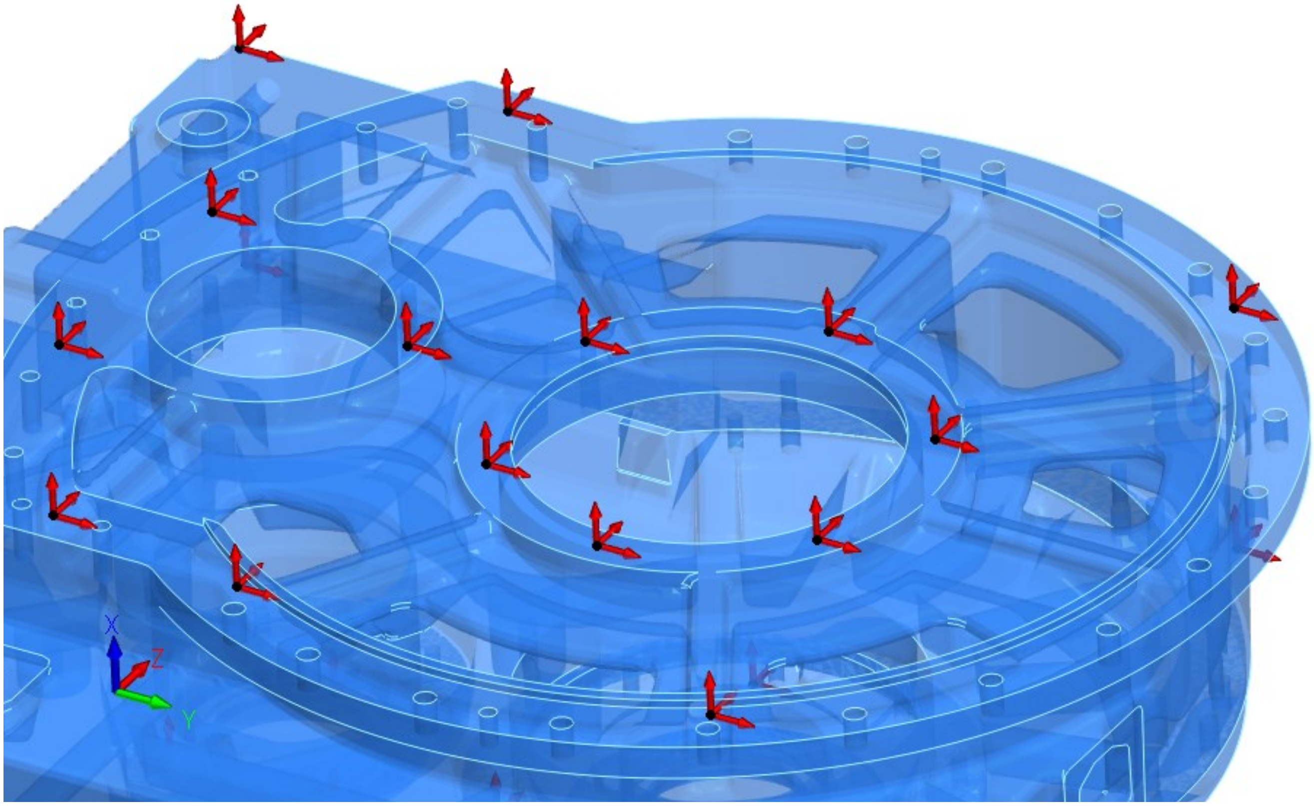
FRF-Based Assembly

- FBA components like boundings and joints to connect FRFs that can be rigid or flexible.
- Support for impedances (viscous damper, structural damping) with local coordinate systems.
- Support for added masses with local coordinate systems, and tuned absorbers.
- Definition of intra-model connectors and multi-point constraints (MPC).
- Definition of FBA forces, enforced displacements and recovery points.
- Computation of internal forces in the FBA connection points (Transfer Path Analysis).



Time Domain Simulation

- Definition and generation of input signals: impulses, sines and random signals.
- Computation of displacements, velocities and acceleration response signals.
- XY plots of signals.
- Export of response signals for pretest analysis.



FEMtools Pretest and Correlation

Advanced Solutions for Modal Pretest Analysis and Test-Analysis Correlation

What is FEMtools Pretest and Correlation?

The FEMtools Pretest and Correlation configuration combines tools for modal test planning and to correlate two sets of data.

If a baseline finite element model is available, then this model can be used to simulate tests. This provides test engineers with optimal locations and directions to excite and suspend the structure, and to position measurement transducers. The FE model can be reduced and converted into a test model.

Correlation analysis requires the quantitative and qualitative analysis of static or dynamic response data from two different sources. Global correlation is used to identify matching shapes. Local correlation is used to find updating variables or to identify structural damage.

These are essential tools for structural dynamics analysts that need to validate and update finite element models, design optimal test conditions, evaluate different modeling strategies, identify modeling errors, detect structural damage, …

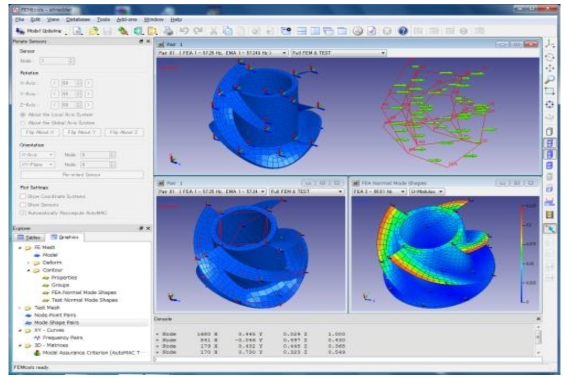
Results from correlation analysis are used to define targets for FE model updating.

Another application is to provide the analyst with information that can only be measured. An example is modal damping, used in modal superposition methods. Modal damping can be obtained experimentally and applied to the analytical mode shape that, using correlation analysis, was found to best match the experimental one.

Modal correlation analysis is also used to scale test mode shapes obtained by output-only modal analysis. The same scaling as used by the analytical mode shapes (e.g. unity modal mass), can be applied to the correlated test modes.

Unlike global correlation analysis, spatial methods can be used to identify areas of better or poorer correlation, which when linked to structural information, can be interpreted in terms of 'modeling error'. Depending on how these tools are used, the results help with the selection of updating variables (parameters), or are used to assess structural damage.

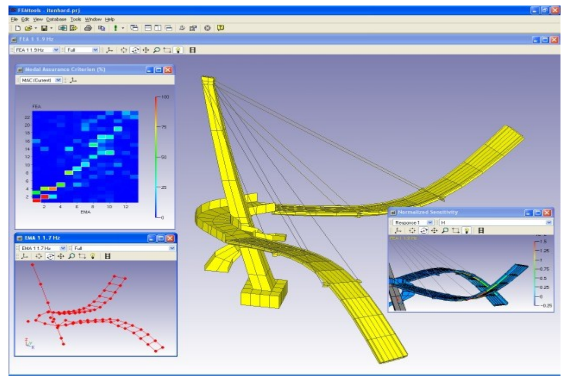
The FEMtools Pretest and Correlation configuration adds the following modules to the framework:



Modal Pretest Analysis

The following tools are included:

- **Target mode selection** - Select modes in the frequency band of interest based on energy considerations.
- **Selection of candidate sensor locations** - Use criteria like accessibility, cost, geometry (surface, edge or corner nodes) or any other user-defined criteria to select candidate locations. A set of fixed sensors can be added.
- **Manual and automated sensor location selection** - Use various criteria to manually or automatically select optimal measurement locations and directions from the selected candidates.
- **Selection of excitation and support locations** - Use various criteria to manually or automatically select optimal excitations and support locations and directions.

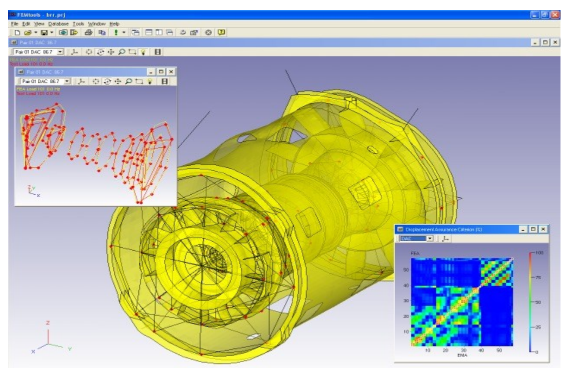


- **Mass Loading Evaluation** - Evaluate the effect of transducer mass on the modal parameters.
- **Creation and Export of Test Model** - Convert an FE model to a test model and export to modal test software. Automatic generation of tracelines. Compute local measurement directions.

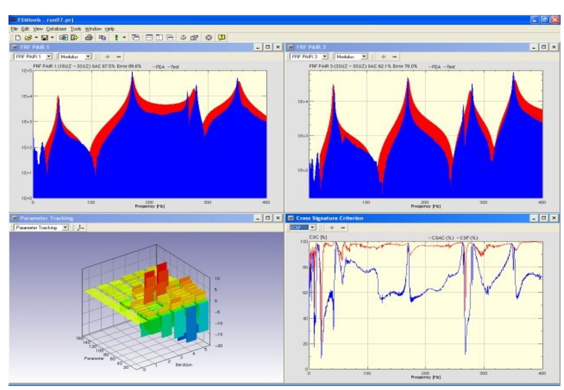
Correlation Analysis

The following tools are included:

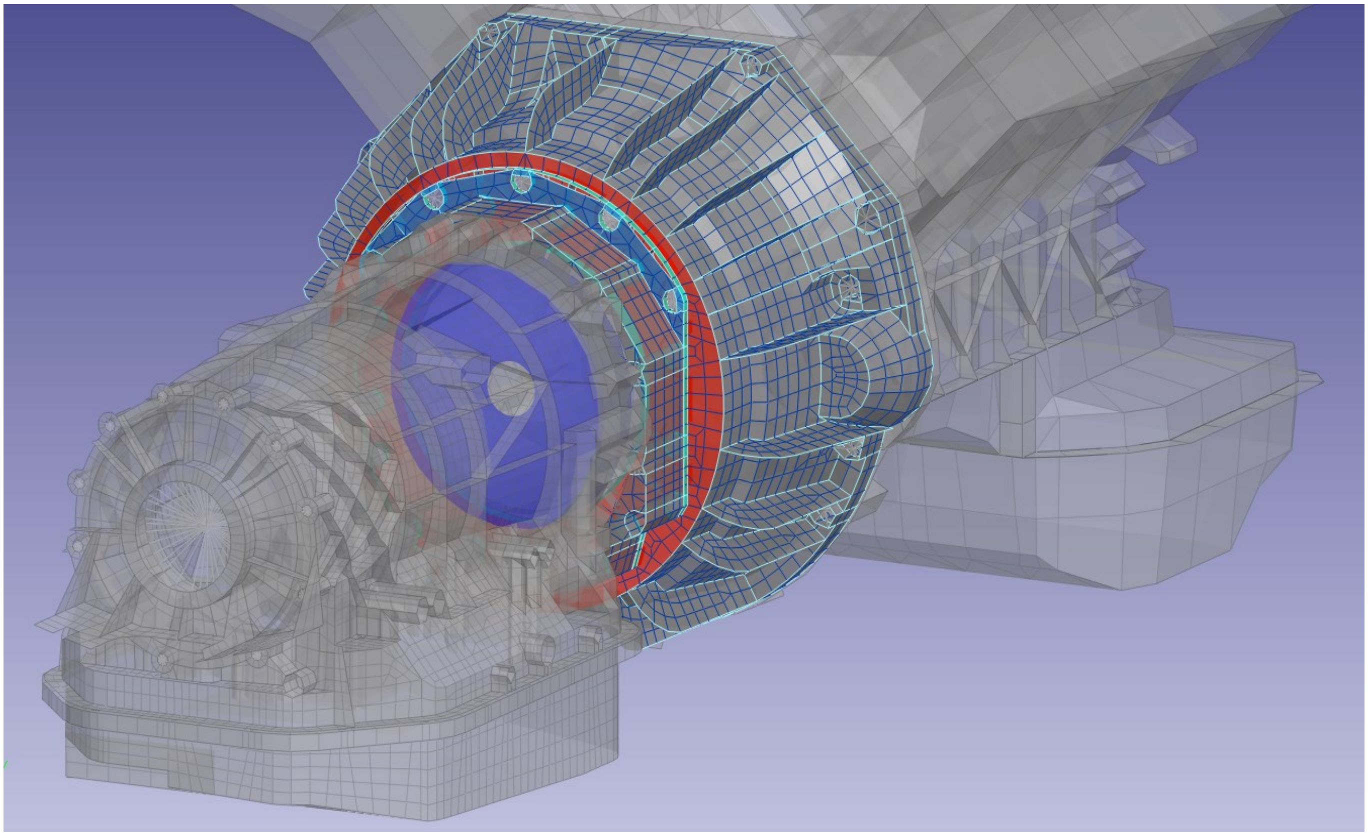
- **Spatial correlation** - Compares location in space between response locations resulting in a table with mapped degrees-of-freedom. This may require changing orientation and scaling of the models, which can be done in a manual way or using automatic tools.
- **Visual shape** correlation - Visually compare shapes (static displacement shapes, mode shapes and operational shapes) using side-by-side, overly and animated displays.



- **Global shape correlation** - Globally compares shapes using various criteria. The result is used for shape pairing.
- **Local shape correlation** - analyzes local spatial correlation between shapes. Results can be interpreted to localize modeling deficiencies and serve as guideline for selecting model updating parameters.
- **Shape** pairing - Creates a table of shape pairs (static, modal or operational).
- **FRF** pairing - Creates a table of FRF pairs.
- **FRF correlation** - Analyzes correlation between FRF functions, either globally between 2 functions or shape and amplitude correlation functions for a set of FRF pairs as function of frequency.
- **Correlation coefficients** - Calculates values of error functions from a selection of reference responses. These functions are used in model updating to monitor the distance between the updated model and a reference.



The modules included with FEMtools Dynamics are also included with FEMtools Pretest and Correlation. Additional tools for interfacing with FE programs or to process test data can be added.



FEMtools Model Updating

An Integrated Solution for Structural Dynamics Simulation, Model Verification, Validation and Updating

Engineering simulation has become an essential tool for supporting decision-making in product design, qualification and certification. To guarantee the required confidence in results obtained with tech-nologies like finite element analysis, these simulation models must be assessed for their predictive capabilities.

Engineers are facing new responsibilities and challenges to comply with new Quality Assurance standards like ASME or ISO 9001 that specify more stringent requirements to demonstrate validity of simulation engineering. This requires the adoption of new workflows for Verification and Validation (V&V) using physical tests or high fidelity models, and in most cases, the updating of input parameters.

Today this workflow can be largely automated thanks to integrated software tools like FEMtools Model Updating that rely on the combined use of simulation and test data.

They can also be used in the framework of the Digital Twin concept and enable applications like Structural Health Monitoring (SHM).

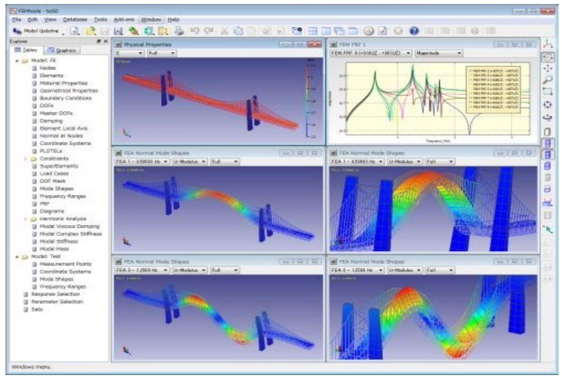
What is FEMtools Model Updating?

FEMtools Model Updating is a toolbox for the validation of a finite element model by using reference data in the form of test data or another high fidelity simulation model.

By including parameter identification, model validation and updating in design and analysis, information about the model and the required level of refinement is obtained in a more efficient and productive way than costly trial-and-error approaches.

FEMtools supports analysts and test engineers in this knowledge and decision-based process, offering a unique collection of tools that complement the general-purpose finite element analysis codes and test systems.

The FEMtools Model Updating configuration adds the following modules to the framework:



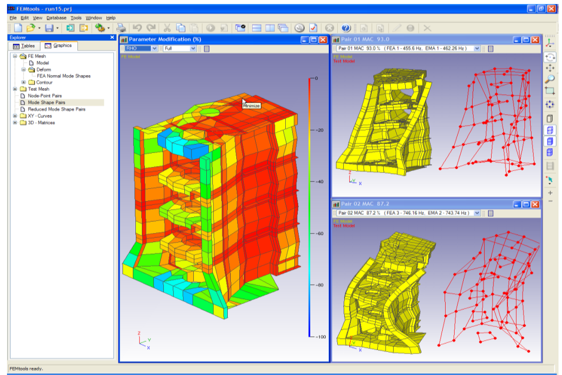
Sensitivity Analysis

Shows which parametric changes are effective and which are not. Supports boundary conditions, material properties, spring stiffness, and element geometry. Sensitivity information is used for design improvement and model updating.

Automated FE Model Updating

Iterative process that modifies updating variables in order to minimize the 'distance' between test and simulation results. Results are used for mesh refinement, or identification of materials, joint stiffness and equivalent properties for use with low fidelity models.

Uncertainty Quantification and Propagation



Probabilistic methods to predict random properties of responses from random input properties.

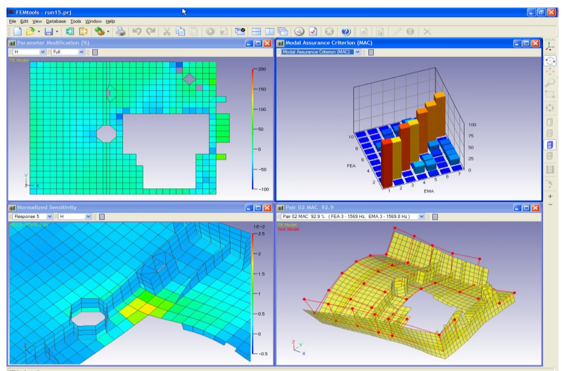
Design of Experiments/Response Surface Modeling

DOE techniques sample the design space in an efficient way with a minimum number of sampling points. RSM is used to build an approximate model from the DOE runs. This model can be used for model updating in a more efficient way than the model from which it is derived.

Force Identification

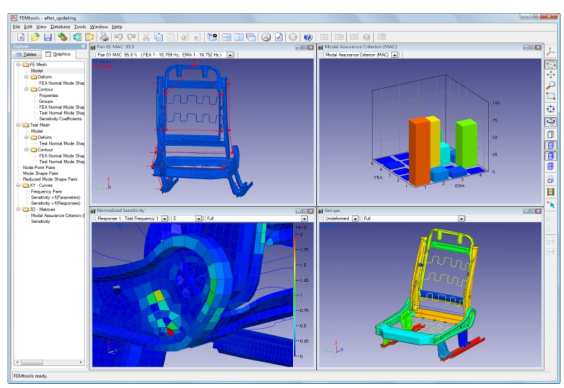
Identification of harmonic forces from measurement of harmonic responses using an updated finite element model.

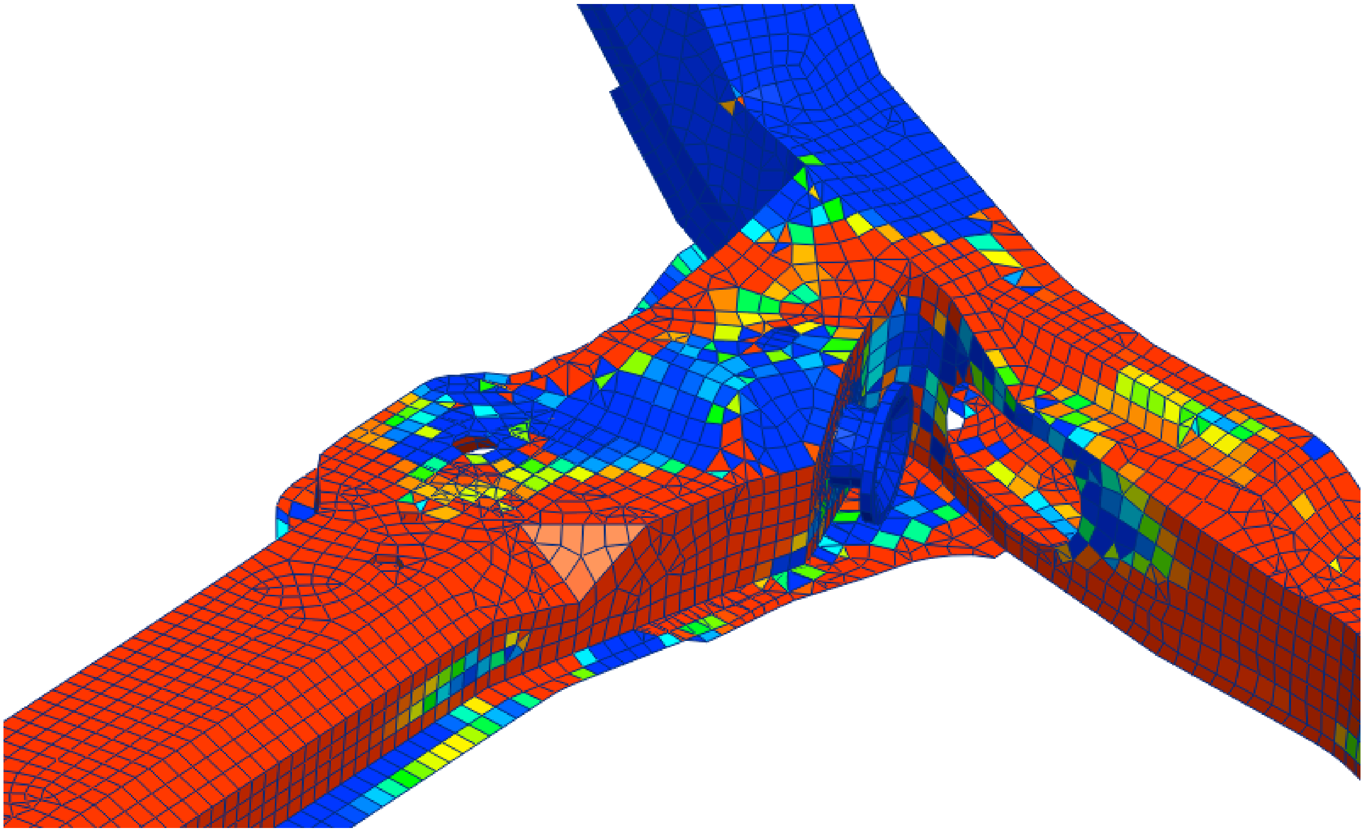
FEMtools Model Updating can be used for



- What-if analysis
- Variational analysis
- Model validation and refinement
- Material Identification
- Identification of structural parameters
- FE Model reduction
- Creation of Digital Twins
- Structural health monitoring and damage identification

The modules included with FEMtools Dynamics and with FEMtools Pretest and Correlation are also included with FEMtools Model Updating. As a result, all tools needed to correlate between two models, validate modeling assumptions, update input parameters and analyze uncertainty are included. Additional tools for interfa-cing with FE programs or to process test data can be added.





FEMtools Optimization

An Integrated Solution for Structural Design Optimization

What is FEMtools Optimization?

FEMtools Optimization is a toolbox for general purpose and structural design optimization.

In combination with numerical optimization algorithms, the finite element analysis method can be used to optimize existing designs or even propose radical new designs. However, a successful application of this approach requires a seamless integration of the optimization routines into the FEA-analysis software.

Based on the acting loads, the design constraints, and the required structural behavior, FEMtools Optimization computes the optimal design parameters for the considered component or assembly.

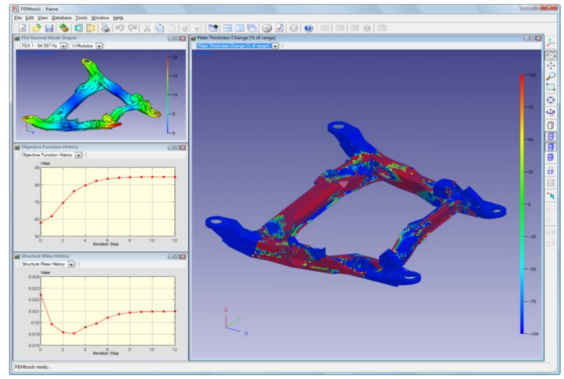
The state-of-the-art optimization techniques of FEMtools Optimization enable to increase the performance of the considered component in a more efficient and productive way than trial-and-error approaches.

FEMtools Optimization has an open architecture providing virtually unlimited flexibility in the problem definition and offering the possibility to drive your preferred FE solver for function re-evaluation or you can use the solvers from FEMtools Dynamics.

Using Validated Models

Combined with FEMtools Model Updating, FEMtools Optimization offers the unique possibility to first validate and update the FE-model of the initial design using test data obtained on a prototype, before optimizing the design of the structure.

General Non-Linear Optimization



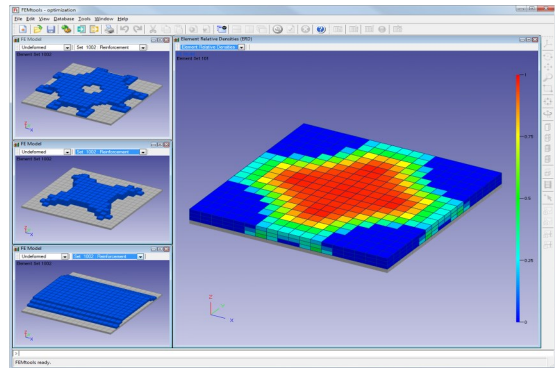
FEMtools Optimization comprises a powerful general non-linear optimization solver. The optimizer has no fixed limits on the number of optimization parameters, objective functions or constraints. The integrated FEMtools Script language can be used to program any arbitrary objective or constraint function.

Sensitivity Analysis

Efficient structural optimization requires the analytical computation of gradient information for the objective and constraint functions. FEMtools Optimization has a sensitivity module that is designed to compute gradients with respect to design parameters such as plate thickness and cross-sectional area in the most efficient way.

Size Optimization

FEMtools Optimization provides a module to solve general size optimization problems. The database management and sensitivity analysis functions enable a user-friendly definition of size optimization problems.

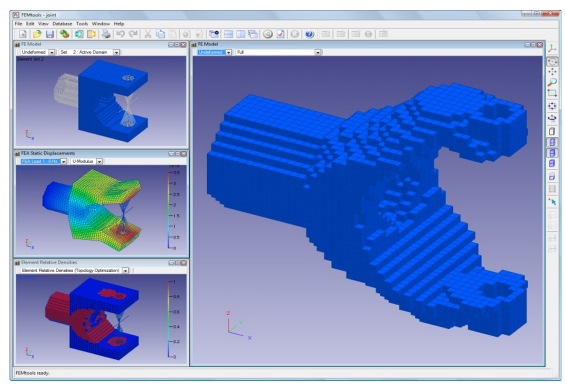


Shape Optimization

Shape optimization problems can be solved using free mesh deformation techniques or using a shape basis. With the first approach the FE-mesh is optimized by shifting the position of a limited number of lattice points that control the shape of the mesh. With the second approach, the final shape is the optimal linear combination of a set of reference shapes. Both approaches do not require the underlying CAD data of the FE-mesh to optimize the shape of the structure.

Topology Optimization

The topology optimization module can create new designs with an optimal topology. FEMtools offers topology optimization for both static and dynamic design problems. A series of commonly used manufacturing constraints are available to improve the manufacturability of the design. Additionally, any other user-defined manufacturing constraint can be added by using the FEMtools Script programming language.

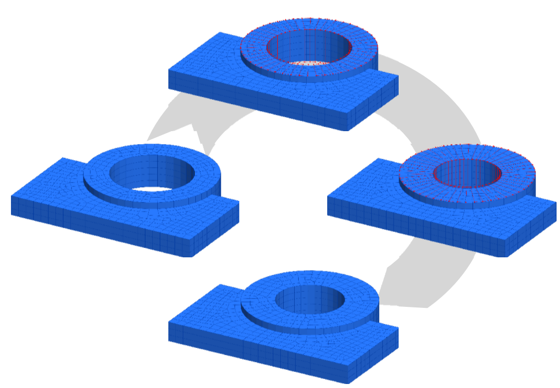


Topometry Optimization

The topometry optimization module can perform a size optimization on an element-by-element basis. FEMtools offers topometry optimization for both static and dynamic problems. A series of commonly used manufacturing constraints are available to improve the manufacturability of the design. Additionally, any other user-defined manufacturing constraint can be added by using the FEMtools Script programming language.

Design of Experiments/Response Surface Modeling

DOE techniques sample the design space in an efficient way with a minimum number of sampling points. RSM is used to build an approximate model from the DOE runs. This model can be used for optimization in a more efficient way than the model from which it is derived.



FEMtools Add-on Modules

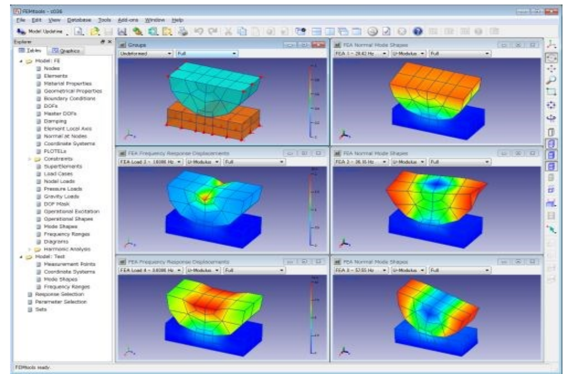
Any FEMtools configuration can be complemented with add-on tools for integration with standard commercial FEA programs or for additional specialized functionality.

FE Data Interfaces and Drivers

The ability to exchange data with other programs is an essential feature of FEMtools. Direct interface programs with standard FE programs are provided so that FEMtools can be integrated in a CAE workflow.

Drivers automate import, export and re-analysis of updated or optimized FE models.

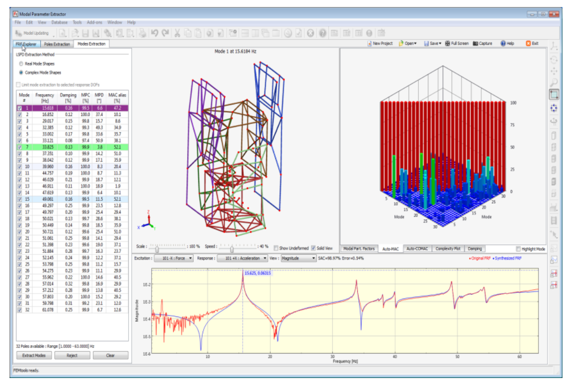
Interfaces and drivers are available for Abaqus, Ansys, LS-Dyna, MSC.Nastran, NX Nastran, OptiStruct and SAP2000. Contact us for availability of interfacing solutions with other programs.



FEMtools Modal Parameter Extractor (MPE)

The MPE tool is used for extracting modal parameters (natural frequencies, mode shapes and modal damping) from a set of Frequency Response Functions (FRFs) or cross-power spectra (XPS) that are computed from response time series.

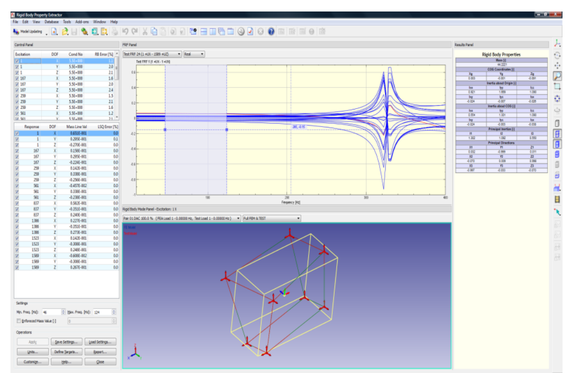
Modal parameters can be used for applications in structural dynamics or to validate and update a finite element model. The extractor can be used interactively or as a component of an automated process for structural health monitoring and evaluation.



FEMtools Rigid Body Properties Extractor (RBPE)

The RBPE tool is used for extracting the rigid body properties of a structure from a set of measured Frequency Response Functions (FRFs). The following properties are obtained from the low-frequency portion of measured accelerances: Mass (M), Center of gravity locations (CoG) and Mass moments of inertia (MoI).

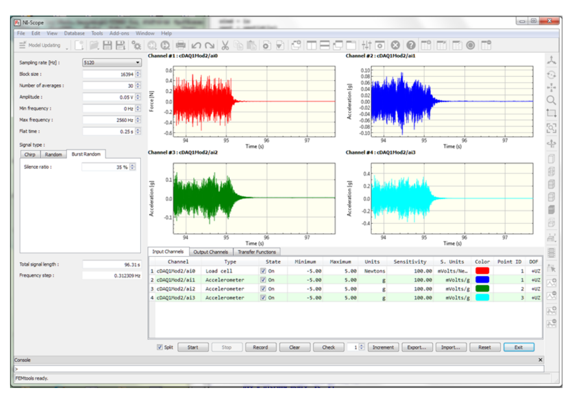
The rigid body properties can be used to validate a finite element model, for substructuring or motion analysis.



FEMtools Data Acquisition (DAQ)

The DAQ tool is used for acquiring multi-channel vibration signals from a structure by utilizing data acquisition hardware from National Instruments to which transducers like accelerometers and an excitation device is connected.

Response spectra are estimated from the vibration signals in real-time and saved for further frequency domain analysis applications. Applications include modal analysis using impact hammer excitation, shaker excitation or operational modal analysis.



General Information

Supported Platforms

FEMtools is available for the following platforms:

- Windows 10, 11 (64-bit)
- Linux (64-bit)

Software Licensing

Flexible nodelocked or network licensing of annual or paid-up licenses.

FEMtools licenses are sold and supported directly by Dynamic Design Solutions and by FEMtools Solutions Partners worldwide. Check the web pages or contact us to find out which partner serves your area.

DDS Customer Services

A dedicated team of specialists at Dynamic Design Solutions continuously improves and extends the FEMtools program. New versions are regularly released. In addition, Dynamic Design Solutions provides the following related services:

- Technology and training courses
- Regular software maintenance
- Technical support by e-mail and phone
- Support website
- On-demand video tutorials
- Custom FEMtools script development
- Custom software development
- Project research

For More Information

Visit www.femtools.com for up-to-date information, data-sheets, technical papers, videos,...

To request specific information via e-mail, send a message to **info@femtools.com**.

About Dynamic Design Solutions

Dynamic Design Solutions ("DDS") is an independent and privately owned company that develops CAE software tools for validating simulation models, optimizing engineering designs, modal testing and automating simulation processes.

We provide solutions that help FEA and test engineers to improve the fidelity of FE models, identify structural properties, assess uncertainty of variables and incorporate variability into models. These validated models can then be used with more confidence to improve performance under real world conditions.

Dynamic Design Solutions is a technology leader in Finite Element (FE) model updating using static or dynamic reference test data. We service a wide spectrum of industries including aerospace, automotive, defense, marine, manufacturing, power, sports and education.

Many prestigious companies and institutions have relied on our unique and proven technologies combined with the best possible support and technical expertise by a dedicated team of specialists.

Dynamic Design Solutions is headquartered in Leuven, Belgium.

www.femtools.com

FEMtools
simulate > validate > update > optimize

Dynamic Design Solutions
Dynamic Design Solutions (DDS) NV
Ambachtenlaan 14/5
B-3001 Leuven
Belgium

+32 16 40 23 00
info@femtools.com
www.femtools.com