FEMtools™ Framework

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

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 ABAQUS, ANSYS, LS-DYNA, NASTRAN, 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.

Key Features

- Convert finite element and test data into a uniform, structured relational database
- Analyze, visualize, manage and report your engineering simulation and test data
- Solve for static and dynamic responses using built-in solvers or by piloting external solvers
- Use FEMtools Script with hundreds of built-in math and API functions to access all data and create your own functions and programs

Applications

FEMtools Framework is a software-neutral CAE application development tool mainly used for developing tools that exploit hybrid test and analysis databases. It provides the foundation for all FEMtools products and for products developed by independent partners.

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

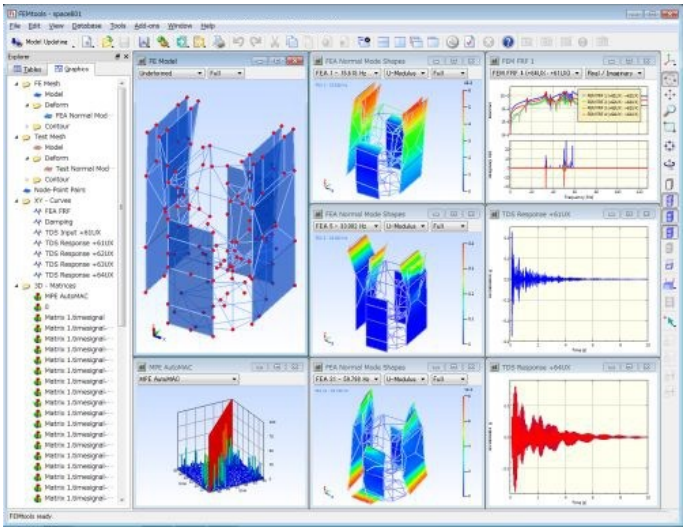
Benefits

- Faster and more efficient development
- Re-use of standard components like graphics viewers and data translators
- Customized user interfaces
- Solver-neutral integration with virtually every FEA package
- Integration with virtually all types of test data
- Computing and OS platform-independent applications
- The programming language and API provide a layer between the database and applications. This reduces the risk of corrupting the database

Framework Components

Data Interfaces and Management

- Optional integrated direct, bi-directional interfaces FEA : ABAQUS, ANSYS, LS-DYNA, NASTRAN, … Test: Universal File, Custom file formats
- Direct import and export of data tables (Excel, Matlab, …)
- Database explorer using tree-lists
- No limitation on FE model size
- Spreadsheet-style table editing
- Conversion of engineering units
- Support of local coordinate systems
- Modal database normalization, scaling, truncation and expansion
- Automatic creation of element and node sets based on properties or by graphical picking
- Boolean operations on sets
- Verification of FEA database integrity
- Mesh generation and morphing tools
- Mesh quality analysis commands



Frequency response analysis of a satellite system.

FEMtools
simulate › validate › update › optimize

Parameter and Response Selection

- Selection of all element material properties, geometrical properties, boundary conditions, lumped masses, and damping factors as parameters
- ANSYS and NASTRAN parameter selection for non-standard element properties.
- Selection of mass, static and dynamic displacements, resonance frequencies, modal displacements, MAC, FRFs, FRF correlation functions, strain and stress as responses
- Lower and upper bound constraints
- Definition of parameter relations

Interactive User Interface

- Customizable menus, toolbars and shortcuts
- Complete digital documentation
- Journal and log file for session replay
- Progress indicators, and cancel button
- Report generator (HTML)
- Floating and dockable toolbars
- Multiple graphics, table and editor windows
- Console window for commands and feedback

Graphics Viewers

- XY-curves, matrix and mesh visualization
- Graphical picking of nodes and elements
- Dynamic viewing (rotation, pan and zoom)
- OpenGL graphics rendering
- Color-coded and vector-coded displays
- Animated, side-by-side and overlay plots
- Export of animated shapes as AVI files
- Export graphics to various graphics file formats

FEMtools Script Language

FEMtools Script is a rapid scripting language that is adapted to the needs of simulation and test engineers. Scripts can be used to add, customize or automate about any imaginable task.

- Compiled byte-code for fast execution
- Integrated script editor and developer menu
- Functions for mathematical programming (array operators and functions, sparse matrices, regular expressions, set manipulation, …)
- Functions for integrating with third party tools (exchange of matrices with Matlab or MS Excel, launching and controlling subprocesses,...)
- Functions for user interface development (graphical picking, dialog boxes, explorer,…)
- Script encoding for protecting intellectual property or source code integrity
- Detailed reference and programmer's manuals

FEMtools API Function Library

The FEMtools API is a library of functions that can be used from within script programs to access FEA data or test data, re-use built-in solvers and analysis tools, user interface development, and pilot external CAE or test tools.

Finite Element Library and Solvers

- Internal finite element library (mass, beams, plates, shells, volume elements, damper)
- Isotropic, orthotropic and anisotropic materials
- Constraint equations (MPC, RBE2, RBAR,...)
- Linear static analysis (displacement, strain, stress)
- Normal and complex modes analysis using sparse Lanczos solver
- Complex modes analysis using modal solver.
- Operational displacement, strain and stress
- Use integrated FEMtools or third party solvers
- Upgrade path to FEMtools Dynamics for more advanced dynamics simulations

Options

- Upgrade to FEMtools Dynamics
- Upgrade to FEMtools Pretest and Correlation
- Upgrade to FEMtools Model Updating
- Upgrade to FEMtools Model Updating and Optimization
- FE interfaces and drivers (Ansys, Abaqus, LS-DYNA, MSC.Nastran, Simcenter Nastran, OptiStruct, SAP2000, Universal File)
- Data Acquisition (Add-on)
- Modal Parameter Extractor (Add-on)
- Rigid Body Properties Extractor (Add-on)

Services

- Installation, training and customization
- Support by e-mail, phone and support site
- Custom software development
- Project research

Supported Platforms

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

For more information, contact us at

Dynamic Design Solutions
CAE Software and Services

Ambachtenlaan 14/5
B-3001 Leuven
Belgium
+32 (0)16 40 23 00
info@femtools.com - www.femtools.com

© 2026 Dynamic Design Solutions (DDS) NV. All rights reserved. Specifications are subject to change.
FEMtools is a trademark of Dynamic Design Solutions (DDS) NV. All other brand and product names are trademarks of their respective holders.