

Your Benefits

- Continuous workflow on the Workbench project level
- FEMFAT user interface analogous to ANSYS Mechanical
- Access to the Parameter Manager to use the Design Explorer or other optimization tools
- Properties are defined based on ANSYS Mechanical geometry
- Weld definition directly on geometry and independent of the mesh
- Optional possibility of displaying the calculation results in Mechanical
- Optional possibility of checking the project in the FEMFAT GUI
- Integration of Job File Commands (all FEMFAT functions are accessible)
- Improved data management to store different FEMFAT simulation systems within a project according to a predefined schema of the Workbench data structure

Interface

- ANSYS



FEMFAT software

BY MAGNA POWERTRAIN

Head Office:

Austria
Magna Powertrain
Engineering Center Steyr GmbH & Co KG
Steyrer Strasse 32, 4300 St.Valentin
Helmut Dannbauer
Senior Manager Simulation / Testing

FEMFAT Support and Sales
Phone: +43 7435 501 5300
femfat.support.mpt@magna.com

Sales Offices:

Japan
Magna International, Tokyo
Noriyuki Muramatsu
Phone: +81 3 3548 0310
noriyuki.muramatsu@magna.com

China
Magna Powertrain, Shanghai
Aisheng Tang
Phone: +86 21 6165 1662
aisheng.tang@magna.com

Sales Partners:

Brazil
Virtual CAE
Sao Caetano
Phone: +55 1142 2913 49
femfat.support.brazil@virtualcae.com.br

France, Belgium & Luxembourg
CIMES France
Valenciennes
Phone: +33 3274 1737 4
support@cimesfrance.com

India
Finite to Infinite
Pune
Phone: +91 86 6846 0087
femfat.support.india@finitetoinfinite.com

Italy
EnginSoft
Padova
Phone: +39 4977 0531 1
femfat.support.italy@enginsoft.it

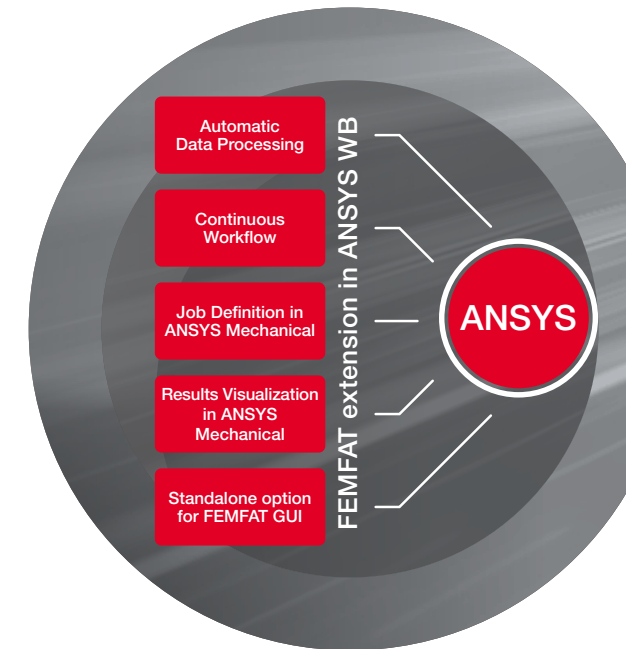
Rep. of Korea
CAE-Cube
Anyang
Phone: +82 3134 43061 2
tjkim@cae-cube.co.kr

USA
Computer Aided Technology (CATI)
Ann Arbor, MI
Phone: +1 734 7863 721 101
rober.baack@cati.com



FEMFAT software

BY MAGNA POWERTRAIN



FEMFAT inside ANSYS

ACT extension for integration of FEMFAT

femfat.magna.com

FEMFAT inside ANSYS

„FEMFAT inside ANSYS“ is an ACT extension for ANSYS Workbench. FEMFAT analyses take advantage of the continuous workflow on Workbench's project schematic and the model tree in ANSYS Mechanical. Fatigue result evaluations can be done directly in ANSYS Mechanical.

With this extension the main functions of FEMFAT are available in ANSYS. With the modules BASIC, MAX, WELD, HEAT and SPECTRAL it is possible to analyse situations from proportional to multiaxial loadings, weld seams, thermal loadings and multiaxial stochastic loadings. The extension can be downloaded from our homepage femfat.magna.com.

Additionally you can find some examples for the first steps of using FEMFAT inside ANSYS extension.

Example

Picture 1 shows a workflow of a Static Structural FE Analysis (System B) connected with a FEMFAT basic simulation (System D). Data transfer in System C checks the compatibility of all FE-simulation types with fatigue analyses. Hence only systems which are compatible with the FEMFAT analysis type may be connected. The FEMFAT job is set up in ANSYS Mechanical. The structure is analogous to the FEMFAT GUI (see picture 2). Optionally a FEMFAT standalone system helps cross-checking the analysis' setup using FEMFAT GUI.

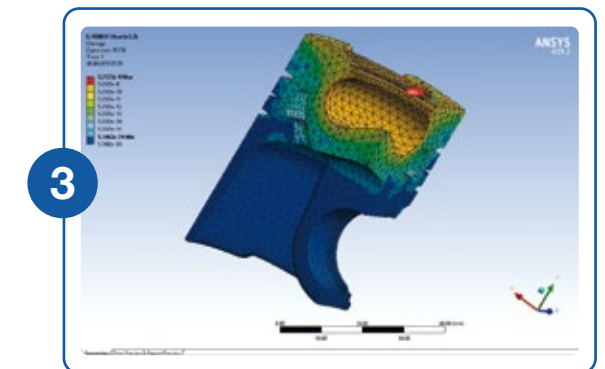
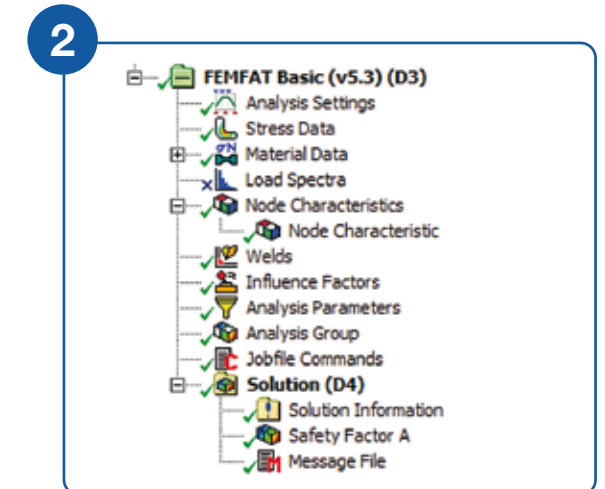
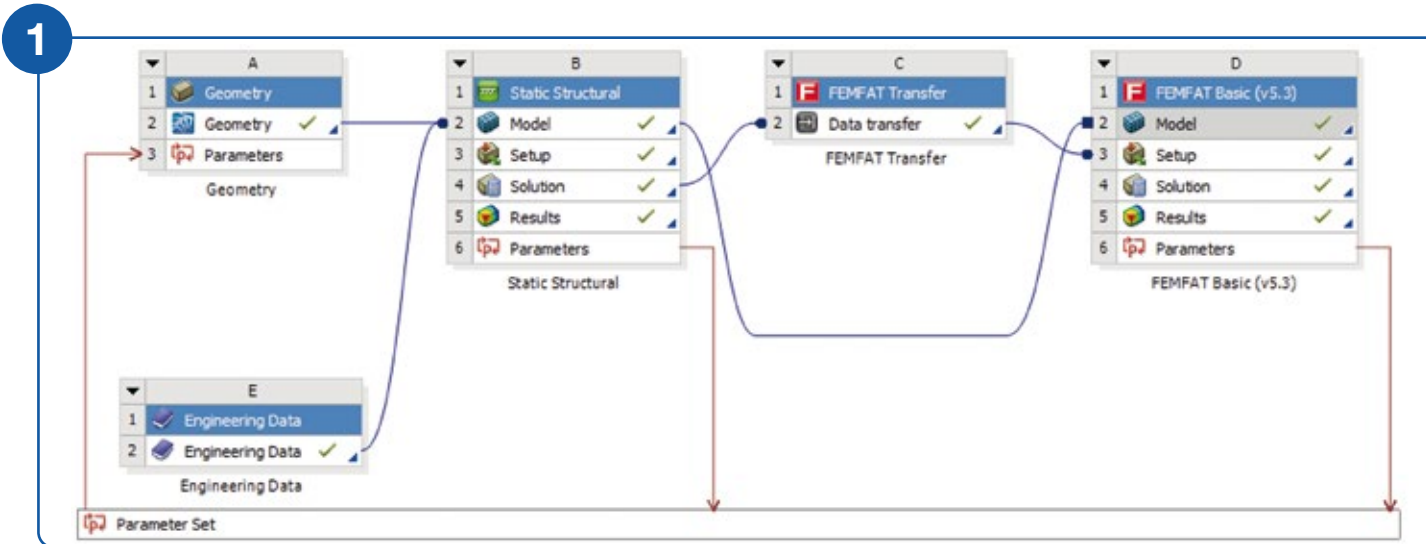
Specialities

- The extension automatically generates the .cdb structure file, even if the FE calculation has already been performed.
- Properties can be defined directly on the CAD structure, alternatively groups (named selections) can be selected.
- Numeric parameters can be set directly in a workflow for the evaluation of Design Points or the use within optimization loops.

Requirements

The fatigue life simulation with FEMFAT inside ANSYS presumes a

- valid FEMFAT basic license (ECS licensing systems LM-X and LSP are supported),
- an extension compatible with your ANSYS Workbench version
- and an associated, compatible FEMFAT version.



FEMFAT result displayed in Mechanical

