

## 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:

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

**India**  
Finite to Infinite, Pune  
Phone: +91 2024 4434 84  
nitin.gokhale@finitetoinfinite.com

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

**USA**  
Caelynx, Ann Arbor, MI  
Phone: +1 734 7863 721 101  
baack@caelynx.com

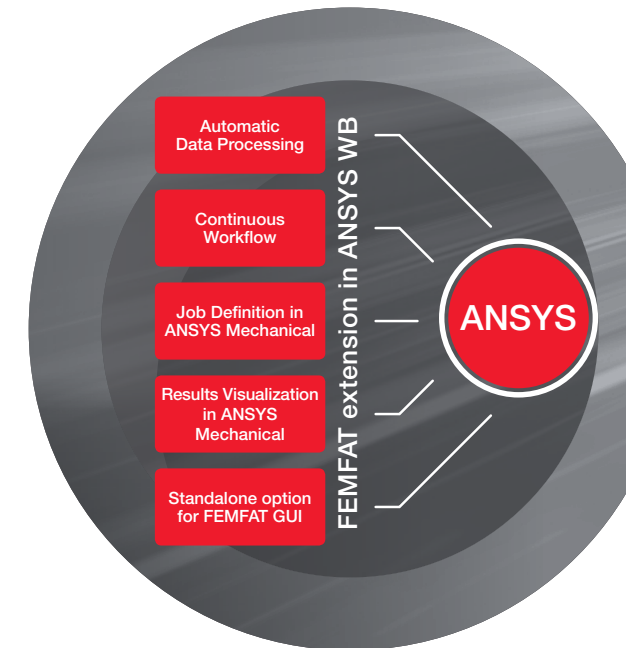
**Italy**  
EnginSoft, Padova  
Phone: +39 4977 0531 1  
femfat.support.italy@enginsoft.it

**France, Belgium & Luxemburg**  
CIMES France, Valenciennes  
Phone: +33 3274 1737 4  
support@cimesfrance.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](http://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.

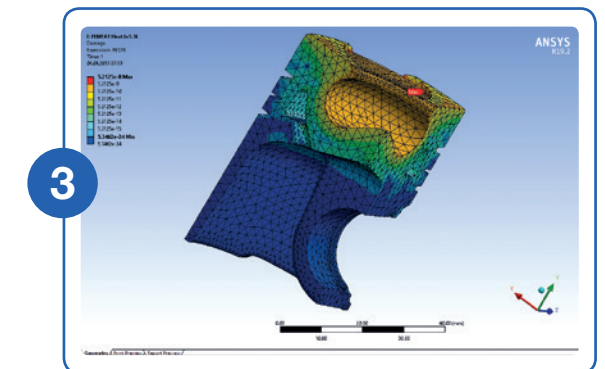
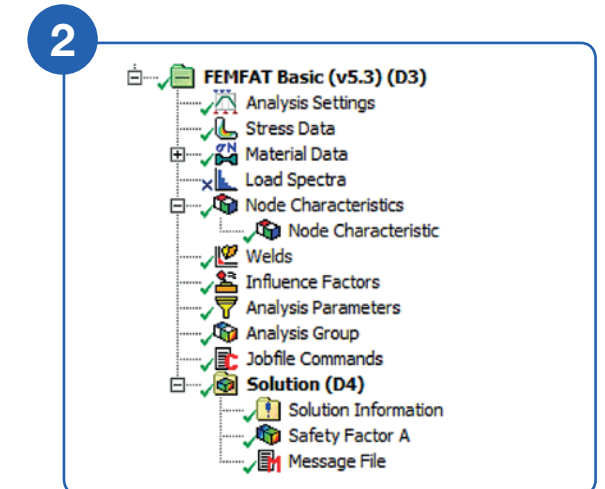
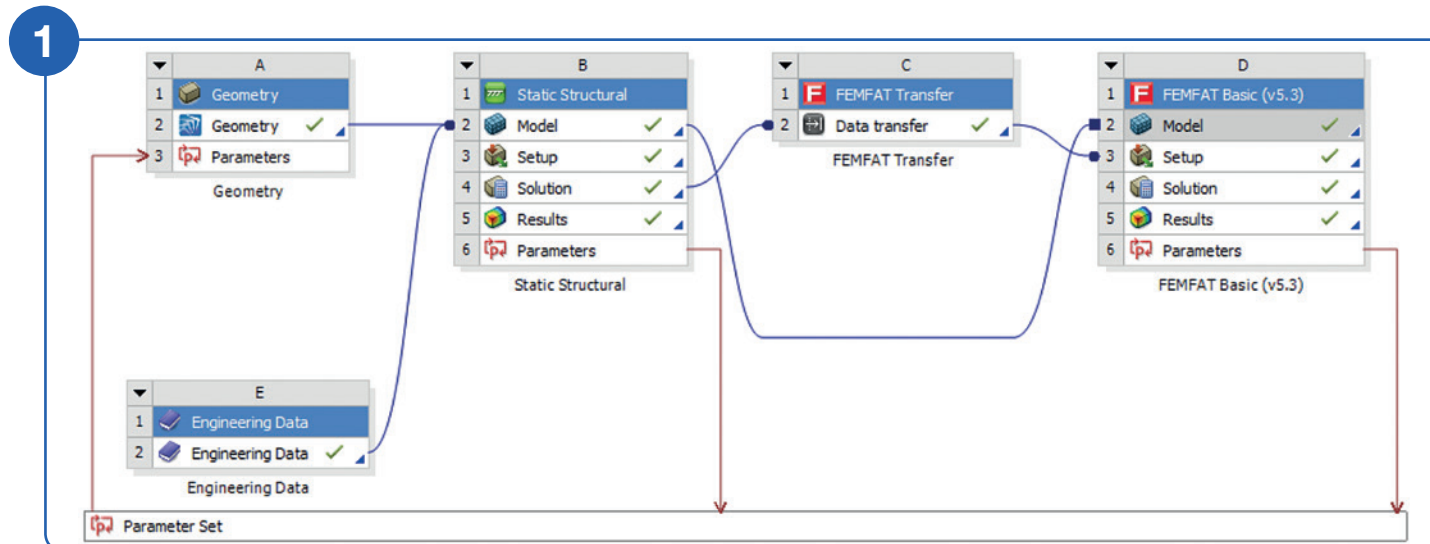
## Specialities

- 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

