Fatigue Analysis of Welded Structures with ANSYS and FEMFAT

ANSYS Conference & 11. CADFEM Austria Users´ Meeting, Linz April 21st 2016

DI Axel Werkhausen, MAGNA/St. Valentin
Introduction

- **ANSYS** and **ANSYS Workbench** are FEM analysis suites – you know maybe better than me – oday it is THEIR Users´Meeting!

- **FEMFAT** is a fatigue solver – some call it a postprocessor only, but there is a solver inside (for the gradient, for the combination of influences, for the equivalent stress from multi axial loading,…)

- Not only a solver- also for some special purpose a pre- and post processing graphic user interface called **FEMFAT visualizer** is available.

- By fruitful cooperation there is an up-to-date interface **ANSYS-FEMFAT** Supporting FEM structure, stresses, nodal temperature, transient stress/strain/temperature, frequency response and transfer function.

- The master-plan is to integrate **FEMFAT** also in the ANSYS Workbench environment (for standard workflow)
Motivation

• Weld seam line assessment for approval according to guidelines is manual exercise and time consuming.
• Weld seam lines in FEM have multiple representation (see next slides)
• The variation in modelling lead to differences in displacements, stress and fatigue life.
• Most realistic 3D-modelling is too much effort and increases the model size tremendously. In 1989 this was not an option.
• Shell modelling was common for large steel sheet structures- so a shell based approach seemed reasonable.
A) 3D solid modelling

• Benefit
  – Realistic FEM-results according local stress (notch stress) approach
  – Takes all weld seam geometries into account

• Disadvantage
  – Increased model size
  – High effort in meshing
  – Not flexible for design changes
  – Not flexible for weld seam type changes
B) Shell model with weld seam inclination

• Benefit
  – Simple FE model (meshing)
  – Fast modelling
  – Takes weld seam size into account
  – More flexible to design changes

• Disadvantage
  – Non realistic distortion, stress and life
  – Not flexible for weld seam type changes
C) Shell model with extra weld seam element

- **Benefit**
  - Simple FE model (meshing)
  - Easy modelling
  - Takes weld seam size into account
  - More flexible to design changes

- **Disadvantage**
  - Needs individual adaption for correct stiffness
  - Not flexible for weld seam type changes
D) Shell model node-to-node

• Benefit
  – Very simple FE model (meshing)
  – flexible for weld seam type changes
  – easy modelling (mid surface)
  – More flexible to design changes

• Disadvantage
  – Not taking asymmetric stiffness into account (bending load)
  – No weld seam size taken into account
Result from investigation

- The optimum to reach should be to combine the benefits from variant A) and variant D) to eliminate the disadvantages.

- **FEMFAT weld** has taken this challenge with a compromise to build up a database for weld seam parameter.

- Since 1989 this database has grown to a high usability dataset.

---

**A**

- **Benefit**
  - Realistic FEM-results according local stress (notch stress) approach
  - Takes all weld seam geometries into account

- **Disadvantage**
  - Increased model size
  - High effort in meshing
  - Not flexible for design changes
  - Not flexible for weld seam type changes

**B**

- **Benefit**
  - Simple FE model (meshing)
  - Fast modelling
  - Takes weld seam size into account
  - More flexible to design changes

- **Disadvantage**
  - Non realistic distortion, stress and life
  - Not flexible for weld seam type changes

**C**

- **Benefit**
  - Very simple FE model (meshing)
  - Flexible for weld seam type changes
  - Easy modelling (mid surface)
  - More flexible to design changes

- **Disadvantage**
  - Needs individual adaption for correct stiffness
  - Not flexible for weld seam type changes

**D**

- **Benefit**
  - Simple FE model (meshing)
  - Easy modelling
  - Takes weld seam size into account
  - More flexible to design changes

- **Disadvantage**
  - Not taking asymmetric stiffness into account (bending load)
  - No weld seam size taken into account
Seam weld analysis in FEMFAT

Weld Type: T-WELD
Seam Shape: One-Side Fillet Weld with Root Under-Cut (Keyhole Notch)

2D Modeling:
- Element type: MAT201 - MAT205
- Equivalent Material: MAT203

Material Sets:
- elset: MAT201
- elset: MAT203
- nset: C100
- nset: C102

Color Codes:
- Color code 100
- Color code 102
Investigate for notch factors in all loading situations.

According to Radaij and Sonsino we investigate in 10mm 2-dimensional submodels for notch factors.

\[ \beta_{LFx} = \frac{\sigma_{notch}}{\sigma_{nominal}} \]
In combination with appropriate Database

- ASCII written
- 54 weldseam types
- easy to extend
- Includes
  - notch factors
  - Survival probability
  - Thickness influence
  - Weldseam strength normal / shear
  - Symbol database
  - Haigh Diagrams
  - S/N curve
  - Temperature influence
Interface ANSYS - FEMFAT

- Material Database: fatigue data, cyclic hardening, Haigh-Diagrams, S/N-curve...
- WELD-Database: acc. Guideline, notch factors, Haigh-Diagrams, S/N-curves...
- Measurement:
  - MBS: *.mcf
  - Load Histories: *.rst, *.lis
  - Load Spectra: *
- FEMFAT visualizer:
  - *.cdb
  - *.fps / *.wdf
- FEMFAT:
  - Nodes Elements PODs Groups
  - Stresses, temperatures,
  - Damage Safety Factors Fatigue Life,...
Data export from ANSYS

- FE geometry from *.cdb-file including set definition
  To obtain this file from Workbench execute the command:
  ```
  CDWRITE, DB, W:\pathname\femfat_ansys, cdb
  ```

- FEM stress result from each loadcase in *.rst-file
  seek in project export directory:
  ```
  workdir
  projectname
  dp0
  SYS
  MECH/file.rst
  ```
Import *.cdb-file
Maximum load is defined as „upper stress“. Leave lower stress at „0“ for pulsating load.
The Tasks:

a) how many cycles does the component withstand until crack initiates?

b) Where does the crack start?

c) Which location in weld seam?
FEMFAT Analysis settings

- **Material:** higher strength weldable steel EN-S355
- **Load spectra:** 1,000 cycles
- **Surface roughness:** 10μm
- **Mean stress influence:** ON (pulsating has mean stress same as amplitude)
- **Damage Analysis:** Miner modified
- **Equivalent stress:** scaled normal stress
- **Analysis filter:** amplitude stress > 40% of fatigue limit (default)
- **FEMFAT weld database:** ECS standard (Radaiji / Sonsino, DIN 15018)
a) Life: 483.4x 1.000 cycles
483.400 cycles
Answer b) Location of crack initiation

- in weld seam (Attribute 100)
- web-element (Type 206, E5652)
- Root crack (NWUTOP = Naht Wurzel TOP-shell side)

Answer c) Seam line direction (From, To)
Weld seam definition in FEMFAT visualizer

- Life demo of weld seam definition in neutralized FE-geometry
- Same results but additional information available from small pictogram
Optional: sensitivity analysis

- See presentation for CADFEM usermeeting in 2014
Thank you for your kind attention!