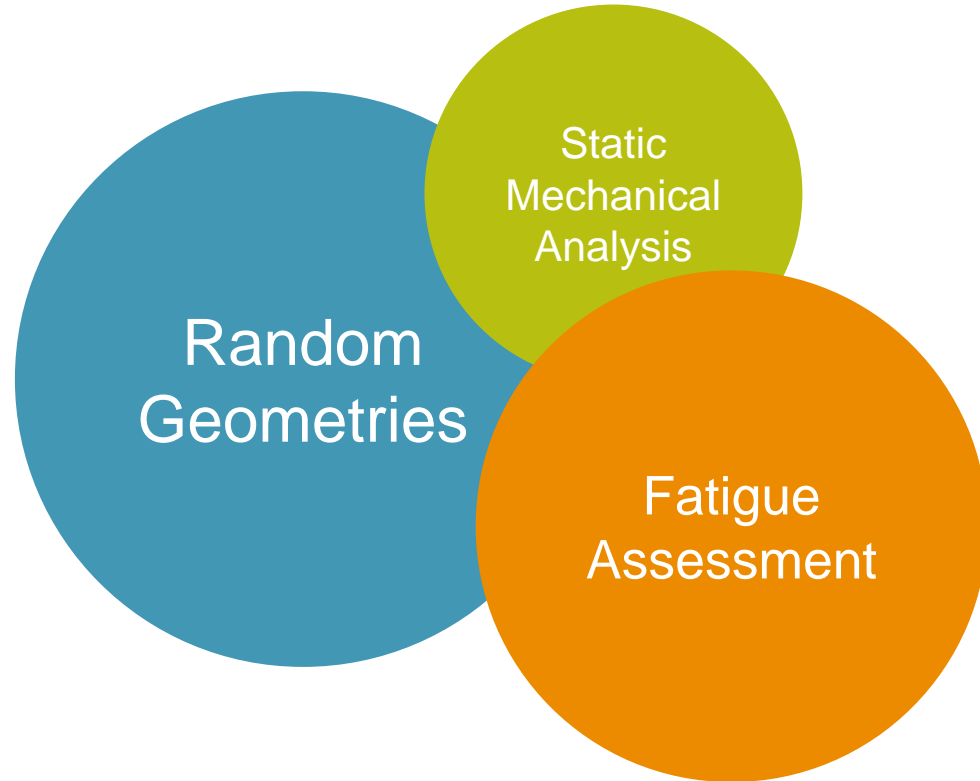




## **Influence of manufacturing tolerances on fatigue life estimation**

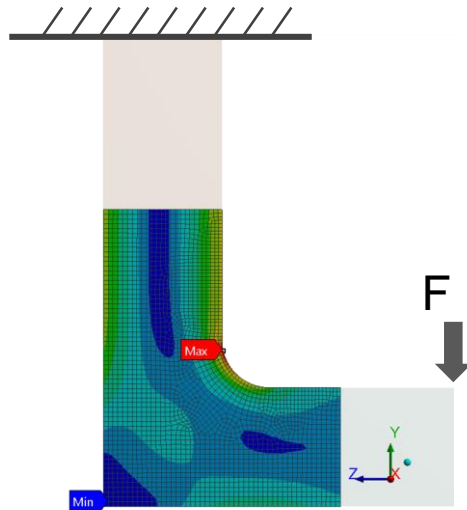
R. Pschera, R. Lampert, S. Wolff



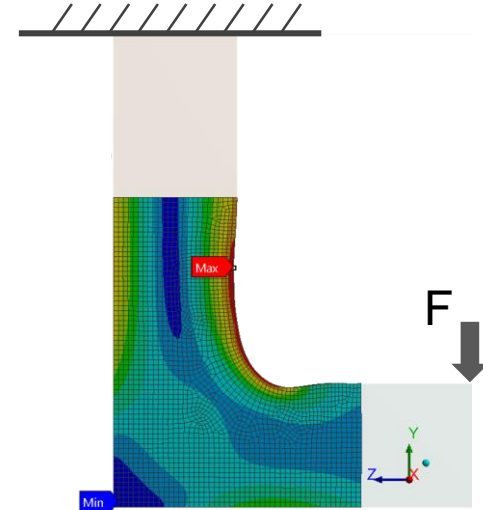
# Problem and Objective

# Geometry optimization can be done easily using optiSLang presuming a managed workflow

Initial System



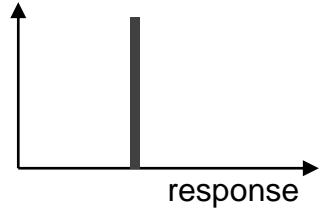
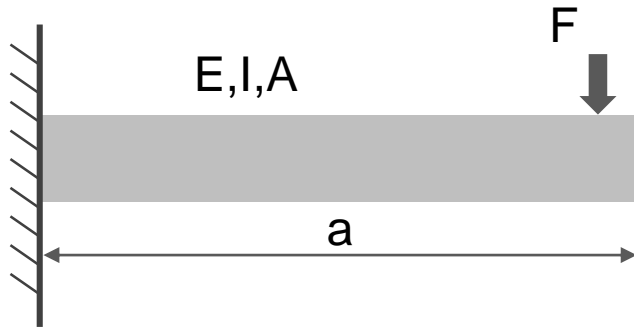
Optimized System



Geometry may be considered as a random variable. Suitable descriptions of their variations is necessary. The simulation process has to be managed in a workflow.

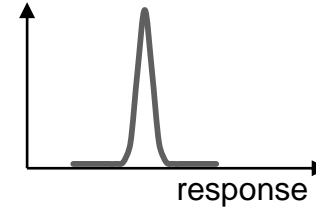
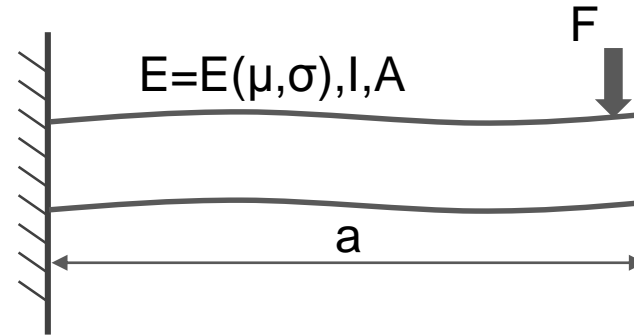
# The stochastic description of a real system presumes a managed workflow and a suitable description of variations

## Ideal system



deterministic response

## Real system



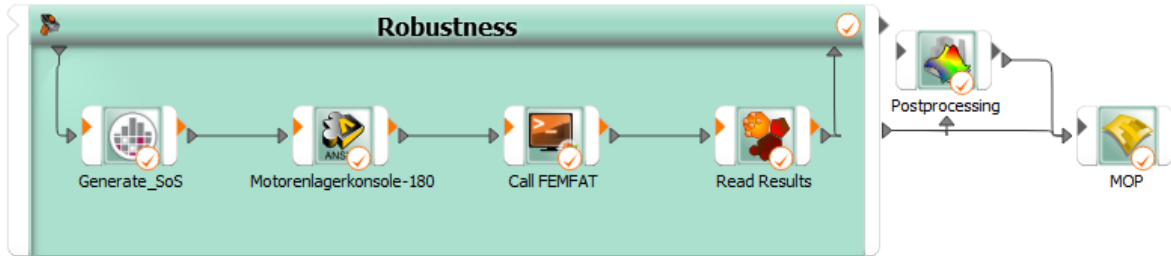
stochastic response



Material properties and geometry may be considered as random variables. Suitable descriptions of their variations is necessary. The simulation process has to be managed in a workflow.

# How to Define the Robustness of a Design

- **Intuitively:** The performance of a robust design is largely unaffected by random perturbations

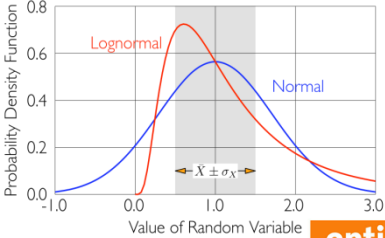


- **Variance indicator:** The coefficient of variation (CV) of the objective function and/or constraint values is smaller than the CV of the input variables
- **Sigma level:** The interval mean $\pm$  sigma level does not reach an undesired performance (e.g. design for six-sigma)
- **Probability indicator:** The probability of reaching undesired performance is smaller than an acceptable value

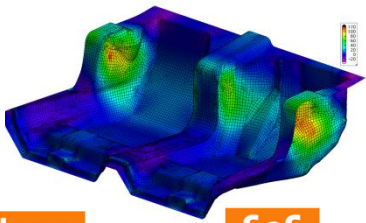
# Robustness Evaluation using optiSLang and SoS



1) Define the robustness space using scatter range, distribution and correlation

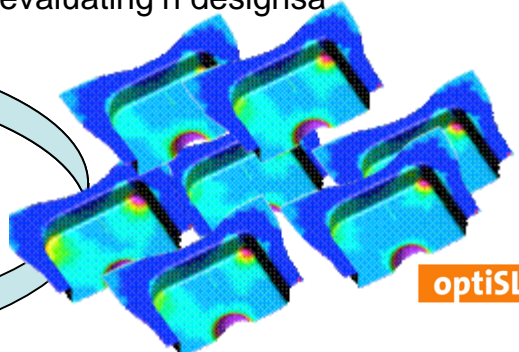


optiSLang



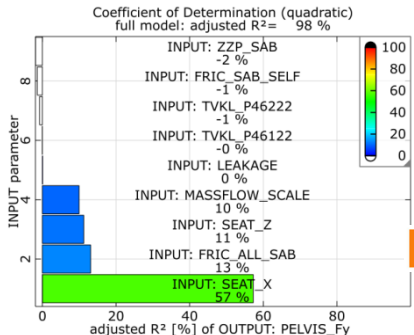
SoS

2) Sampling: Scan the robustness space by producing and evaluating n designs



optiSLang

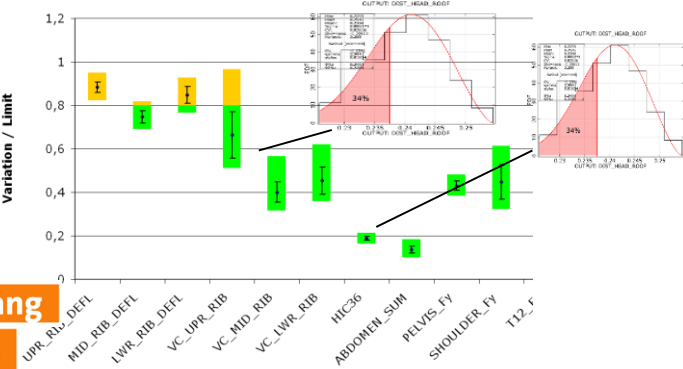
5) Identify the most important scattering variables



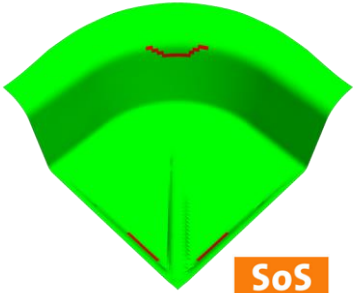
optiSLang

SoS

4) Check the variation



3) Identify statistical hot spots



SoS

# Decision process

## Size of test series vs. numerical accuracy

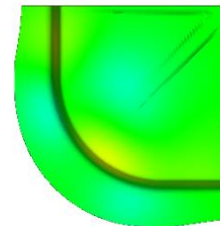
Depending on available data:

**1. No or only a single measurement:**

Use assumptions

Synthetic random field model

Test if the field variations have impact

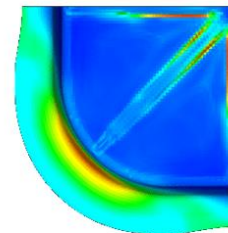


**2. Few number of measurements:**

Assumed correlation, but empirical mean/sigma

Synthetic random field model

Test if “true” magnitude of variation is important

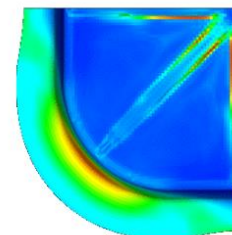


**3. Many measurements:**

Anisotropic, inhomogeneous, Non-Gaussian

Empirical random field model

Model accurate statistics using large test series

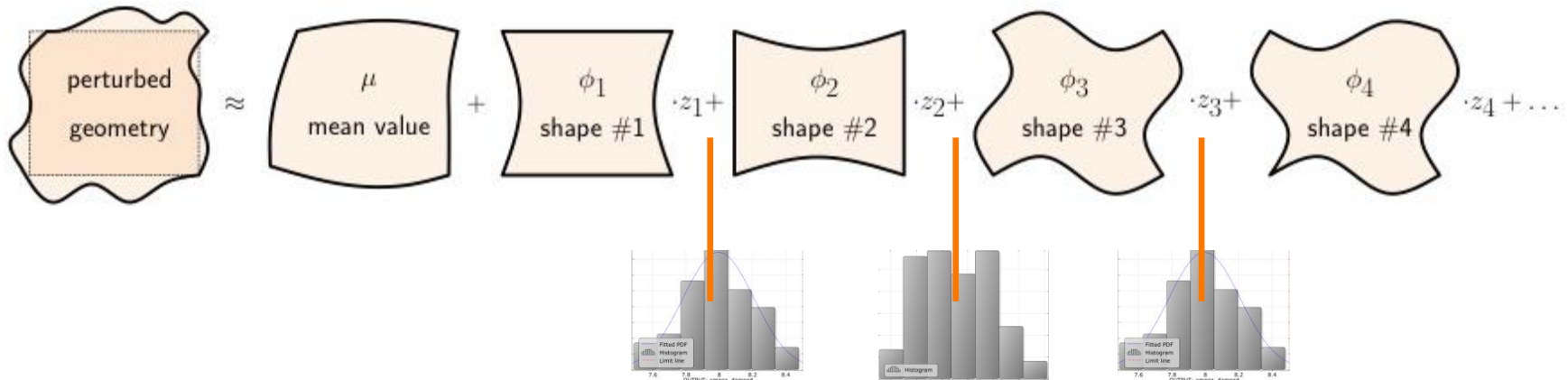




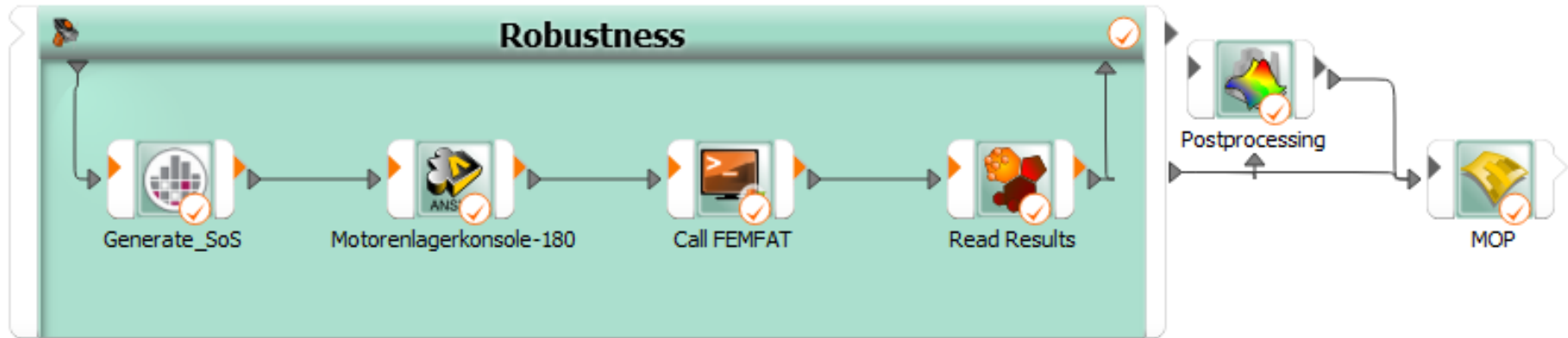
# Parameterization of measured geometries

## Based on measurements or virtual experiments

- Measured spatial field variations:
  - Geometries, thinning, strains, etc.
- Spatial field variations from simulation:
  - Displacements, thinning, remaining stresses, plastic strains, etc.
- New parameters “z”: describe the full variation shape pattern of the measured geometry, strain field, thinning etc.



# A workflow is set up in optiSlang linking SoS, ANSYS WB and FEMFAT to evaluate robustness of the cast structure



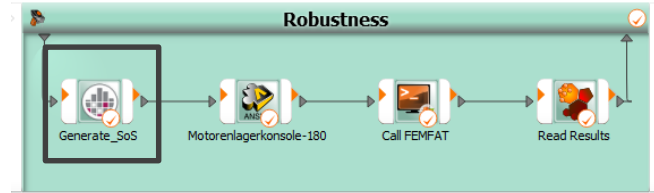
- When material, geometry, process or environmental scatter is significantly affecting the performance of important response values
- When significant scatter of performance is observed in reality
- and there is a doubt that safety distances may be too small or safety distances should be minimized for economical reasons

 Fully automatic transfer of data from the each single task to another

# Random Geometries

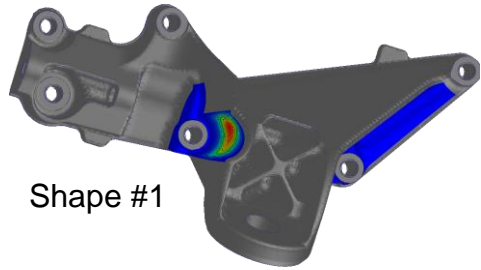
# Generate random geometry with SoS

Parameter	Name	Parameter type	Reference value	Constant	Value type	Resolution	Range	Range plo	PDF	Type
1	amp_disp_node_shape_1	Stochastic	0	<input type="checkbox"/>	REAL	Continuo...				NO...
2	amp_disp_node_shape_2	Stochastic	0	<input type="checkbox"/>	REAL	Continuo...				NO...
3	amp_disp_node_shape_3	Stochastic	0	<input type="checkbox"/>	REAL	Continuo...				NO...
4	amp_disp_node_shape_4	Stochastic	0	<input type="checkbox"/>	REAL	Continuo...				NO...
5	amp_disp_node_shape_5	Stochastic	0	<input type="checkbox"/>	REAL	Continuo...				NO...
6	amp_disp_node_shape_6	Stochastic	0	<input type="checkbox"/>	REAL	Continuo...				NO...
7	amp_disp_node_shape_7	Stochastic	0	<input type="checkbox"/>	REAL	Continuo...				NO...
8	amp_disp_node_shape_8	Stochastic	0	<input type="checkbox"/>	REAL	Continuo...				NO...
9	amp_disp_node_shape_9	Stochastic	0	<input type="checkbox"/>	REAL	Continuo...				NO...
10	amp_disp_node_shape_10	Stochastic	0	<input type="checkbox"/>	REAL	Continuo...				NO...
11	amp_disp_node_shape_11	Stochastic	0	<input type="checkbox"/>	REAL	Continuo...				NO...
12	amp_disp_node_shape_12	Stochastic	0	<input type="checkbox"/>	REAL	Continuo...				NO...

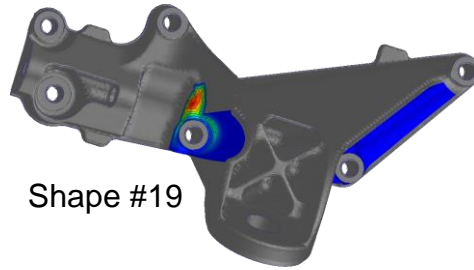


- 20 parameters as coefficient
- For 20 different spatial distributions (shapes) used
- All of them are normal distributed
- Mean value and standard deviation are obtained from analysis of measurements
- Enable displacements into negative and positive directions (Add/Remove material)

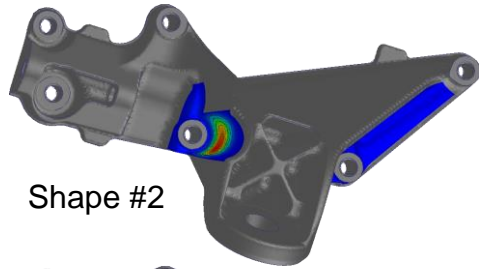
# Generate random geometry with SoS: New models in 3.3.1



Shape #1

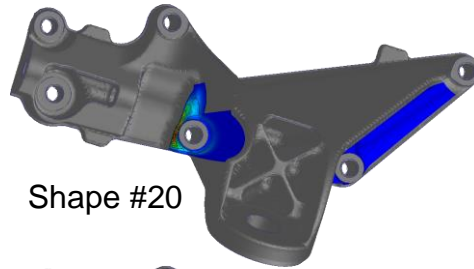


Shape #19



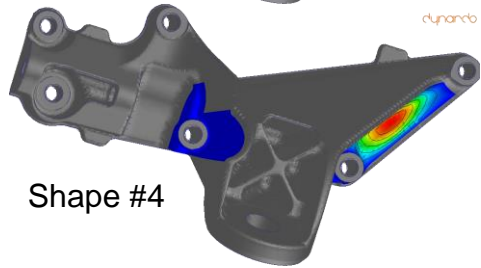
Shape #2

dynareto



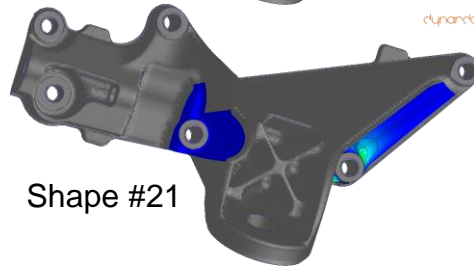
Shape #20

dynareto



Shape #4

dynareto

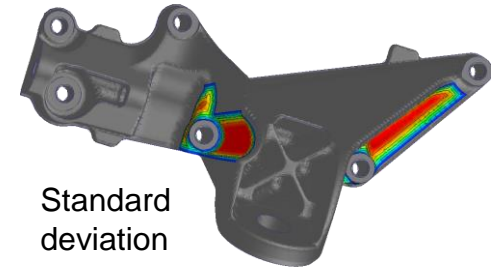
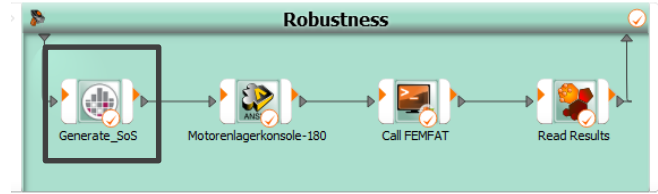


Shape #21

dynareto

dynareto

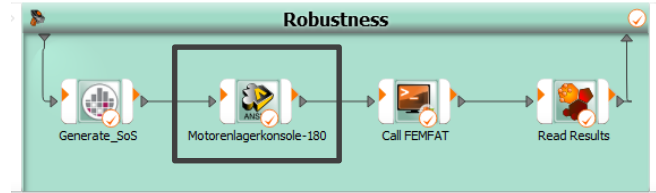
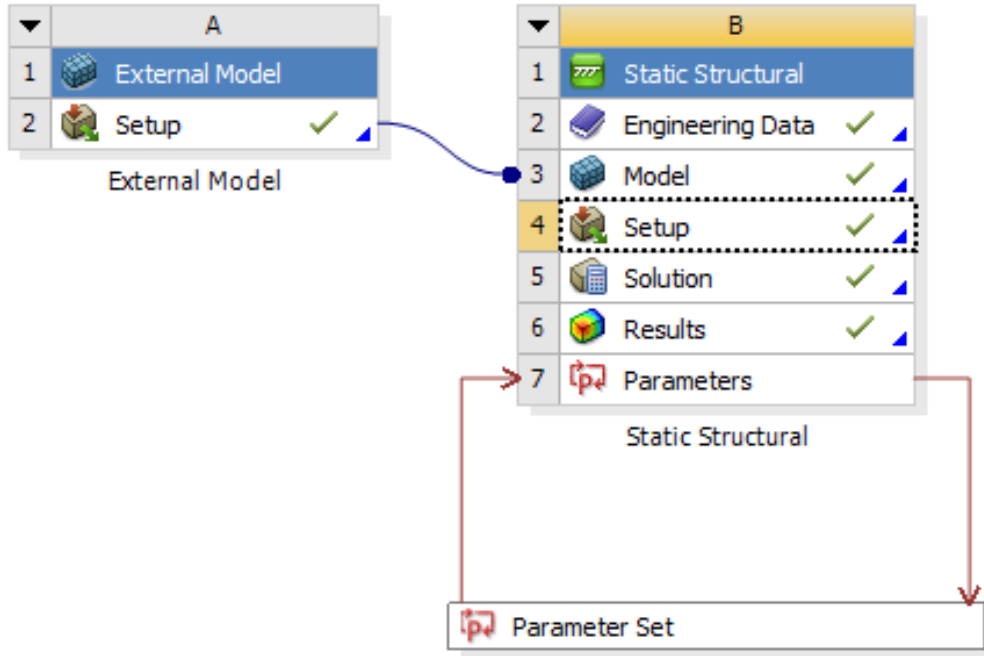
dynareto



Standard deviation

dynareto

# Static Mechanical Analysis



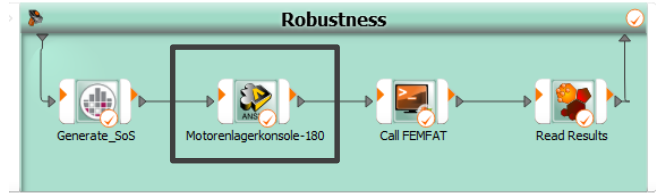
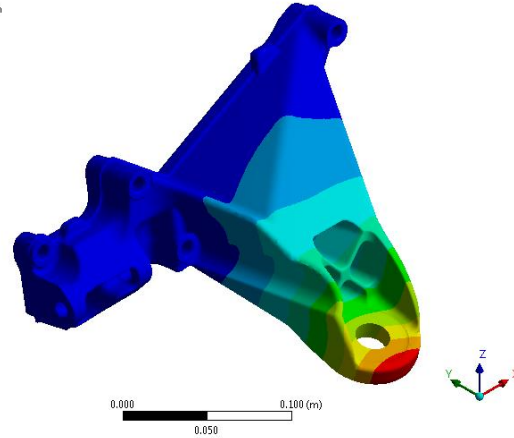
- Geometry is changed inside of the Setup component
- The result file, containing stresses is used for following fatigue evaluation
- Additional results like max. displacements and max. equivalent stresses are considered as responses

# Static mechanical evaluation with ANSYS WB 18.0

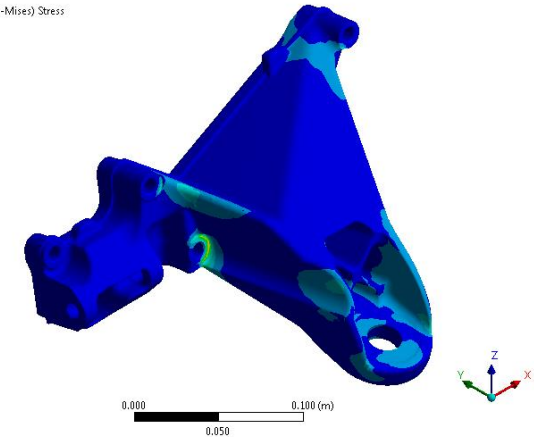
**Project**

- Model (B3)
  - Geometry
  - Coordinate Systems
  - Connections
  - Mesh
  - Named Selections
  - Static Structural (B4)
    - Analysis Settings
    - Nodal Force
    - Nodal Displacement
    - Nodal Displacement :
    - Modify Geometry
  - Solution (B5)
    - Solution Information
    - Total Deformation
    - Equivalent Stress
    - Commands (APDL)

**B: Static Structural**  
Total Deformation  
Type: Total Deformation  
Unit: m  
Time: 1



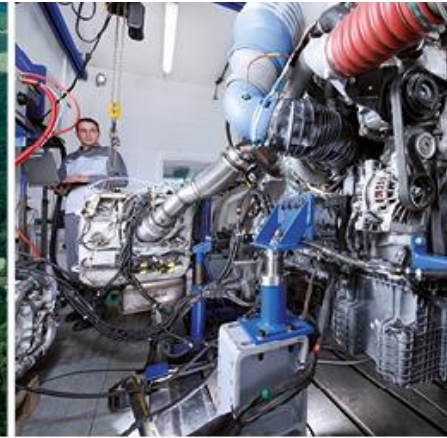
**B: Static Structural**  
Equivalent Stress  
Type: Equivalent (von-Mises) Stress  
Unit: Pa  
Time: 1





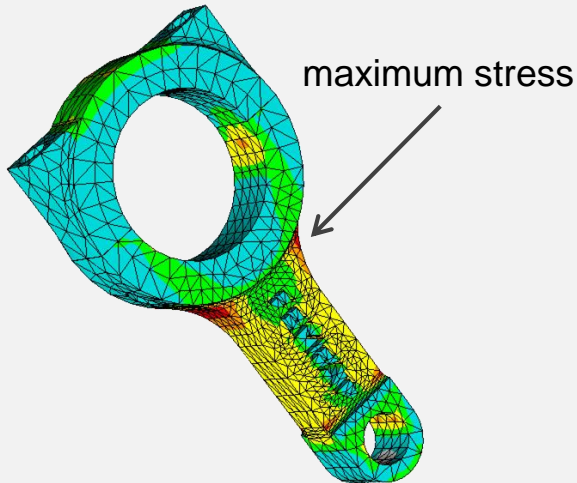
# Fatigue Assessment

FEMFAT is developed in co-operation with computational engineers in our structural analysis department at MAGNA in St. Valentin, Austria.

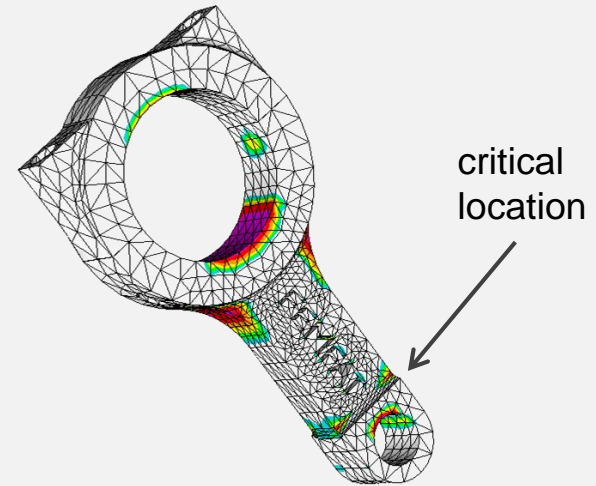


# The exclusive analysis of stress in a traditional way doesn't often reveal damage occurrence at the right point.

## Traditional view



## Modern life-cycle stress analysis



Only modern fatigue analysis tools are capable of predicting critical crack locations and the number of load cycles until failure.

## **FEMFAT** basic

Standard/Minimum configuration; includes all the interfaces and material database, handles 2 stress states plus one assembly load case for life- or safety factor analysis

## **FEMFAT** laminate

A software module for fatigue life prediction on layered infinite fiber reinforced materials

## **FEMFAT** plast

Module to consider the effect of mean stress and/or amplitude stress rearrangement from linear stresses when local plastic deformation occurs using Neuber correction

## **FEMFAT** break

A software module for assessing static safety factors in combination with BASIC or MAX

## **FEMFAT** max

Module for fatigue analysis of MultiAXially loaded components using time histories of loads or series of stress states

## **FEMFAT** visualizer

FAST 3D post-processor to display the FE-model, fatigue results and stresses including a feature to generate animations and 3D pdf files. Unmatched for weld seam definition

## **FEMFAT** heat

For low cycle fatigue analysis of components which are exposed to thermo-mechanical loads (e.g. cylinder heads, exhaust manifolds) and suffer from mechanical, creep and oxidation damage

## **FEMFAT** parallel

Take the advantage to use more than only one CPU of your multi-core workstation to speed up your analysis

## **FEMFAT** weld

Module for fatigue analysis of welding seams for steel and aluminum using notch stress method and standards (DIN 15018, EUROCODE 3 and 9, BS 7608, IIW)

## **FEMFAT** strain

A software module for assessing damage from measured strains and comparing stresses from FEA and testing

## **FEMFAT** spot

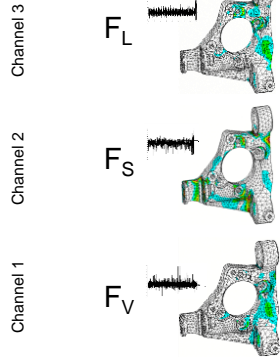
Module for predicting fatigue of spot-joints (welds, rivets) in FE-shell structures

## **FEMFAT** spectral

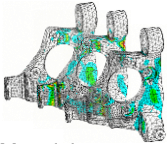
Random response fatigue analysis using PSD (Power Spectral Densities) loads

# Complex loading situations can be assessed in an accurate and efficient way using FEMFAT max

1 Load history for each channel



2 One stress result for each channel

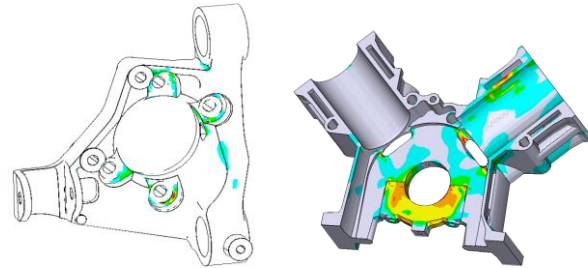


3 Material

Specimen Material Data



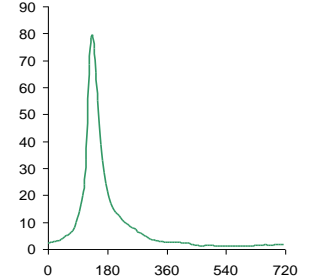
In order to analyze the interaction of all loads, all stress information is superimposed, transformed to an equivalent stress and rain-flow counted. Then the operational strength analysis begins with the help of local S/N curves including relevant influences such as notches, mean stress, isothermal temperature,...



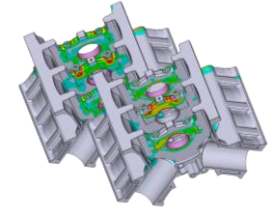
## Results

- damage values,
- safety factors or
- safety factors related to a defined cycle number

1 Transient load condition in time



2 One stress result for each timestep



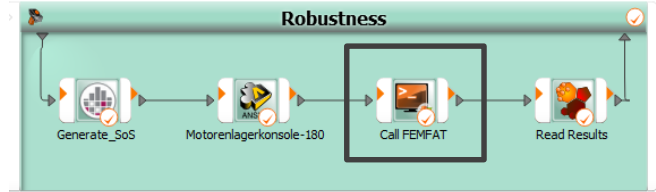
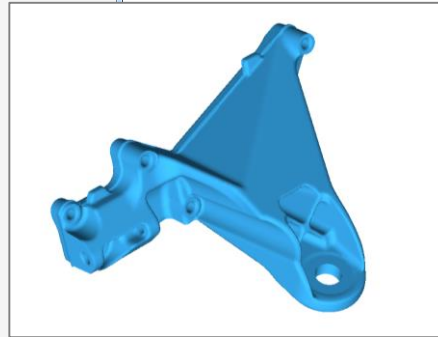
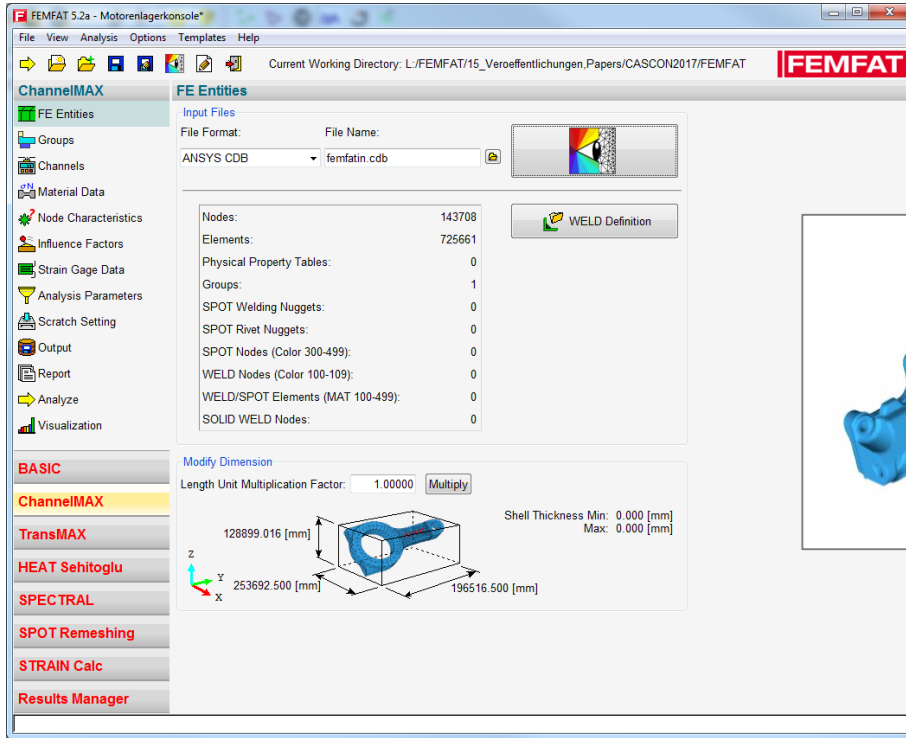
3 Material

Specimen Material Data





# Information of structure is read into FEMFAT using the .cdb interface



# The loading is applied by unit loadcase and the channel history

FEMFAT 5.2a - Motorenlagerkonsole\*

File View Analysis Options Templates Help

Current Working Directory: L:/FEMFAT/15\_Veroeffentlichungen/Papers/CASCON2017/FEMFAT

### ChannelIMAX

FE Entities  
Groups  
Channels  
Material Data  
Node Characteristics  
Influence Factors  
Strain Gage Data  
Analysis Parameters  
Scratch Setting  
Output  
Report  
Analyze  
Visualization

### Channels

Channel Definition

Number of Channels: 3     Auto Fill Anchor  Channel Label: 0  Last

Current Channel: 3

Stress Format Specific Options  
None

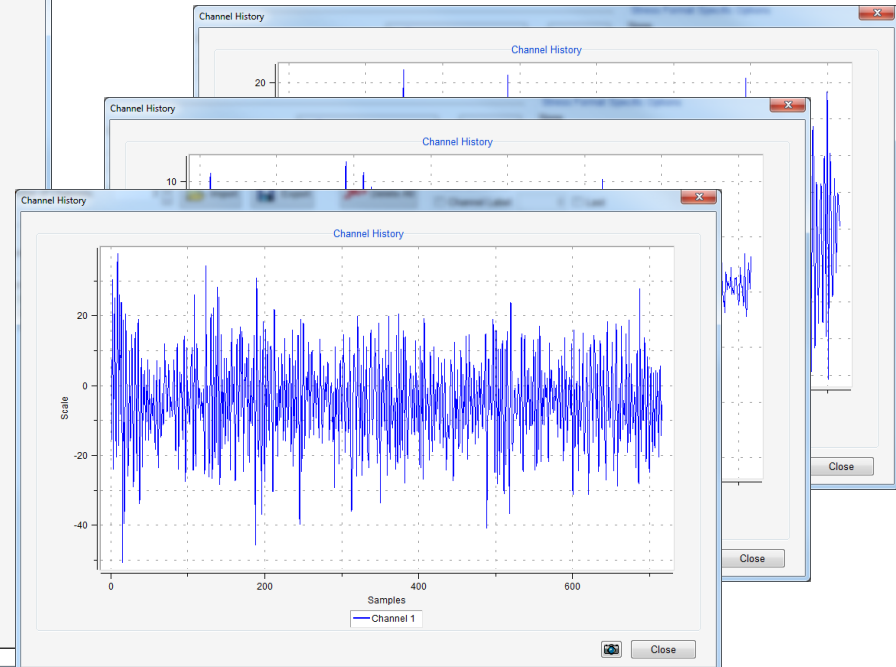
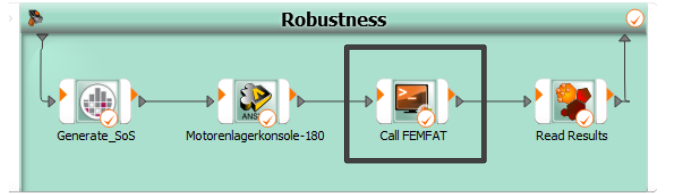
Lbl	Format	Stress File	LC	Factor	L HIST	Load History File	Row	Col	SCR	Scratch File
1	RST AN...	file.rst	1	50000.0	RPC AS...	...nsole_lastzeitverlauf.txt	1	1	ASC	...nlagerkonsole_1.fms
2	RST AN...	file.rst	2	50000.0	RPC AS...	...nsole_lastzeitverlauf.txt	2	2	ASC	...nlagerkonsole_2.fms
3	RST AN...	file.rst	3	50000.0	RPC AS...	...nsole_lastzeitverlauf.txt	3	3	ASC	...nlagerkonsole_3.fms

Loadcase 1

Load History

Number of Datasets: 3  
Number of Samples per Channel: 716

**BASIC**  
**ChannelIMAX**  
**TransMAX**  
**HEAT** Sehitglu  
**SPECTRAL**  
**SPOT** Remeshing  
**STRAIN** Calc  
**Results Manager**



# Material parameters for fatigue analysis are generated in FEMFAT and stored in a .ffd file

**FEMFAT 5.2a - Motorenlagerkonsole\***

File View Analysis Options Templates Help

Current Working Directory: L:/FEMFAT/15\_Veroeffentlichungen/Papers/CASCON2017/FEMFAT

**ChannelMAX**

- FE Entities
- Groups
- Channels
- Material Data
- Node Characteristics
- Influence Factors
- Strain Gage Data
- Analysis Parameters
- Scratch Setting
- Output
- Report
- Analyze
- Visualization

**Material Data**

Manage Materials

- 1 - AISI9Cu3(Fe) F Aluminium
- 1 - AISI9Cu3(Fe) F Aluminium

Material Generator

Controlling:  Stress  Strain

Standard: FKM

Diagrams

- S-N
- H-N
- Haigh

Buttons: Import, New, Export, Copy, Report, Delete, Open, Open

Defect Definition

Buttons: Open

Header Lines

Material and Specimen Name: AISI9Cu3(Fe) F Aluminiumgusswerkstoff DIN EN

Remarks: EN AC-46000, Herstellungszustand

Data Source: Source: FKM 2002 & Material Generator MATG

General Data

- Linear Static Data
- Multilinear Young's Modulus (not used for analysis)
- Cyclic Stabilized Data
- Strength Data
  - Tension
  - Compression
  - Bending
  - Shear
- Header Lines of S-N Data
- General S/N-Data
- Type Dependent S-N Data
- Damage Accumulation (optional)
- Casting Data (optional)
- User defined Haigh Diagram (optional)
- Properties at High Temperature (optional)
- Defect Definition (optional)

**BASIC**

**ChannelMAX**

**TransMAX**

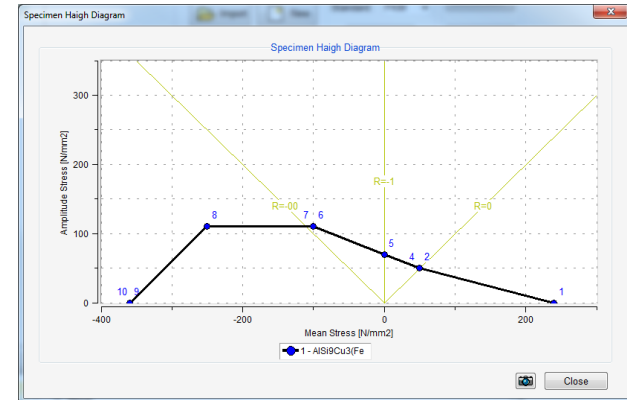
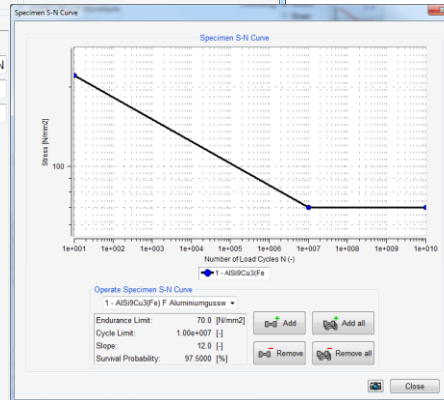
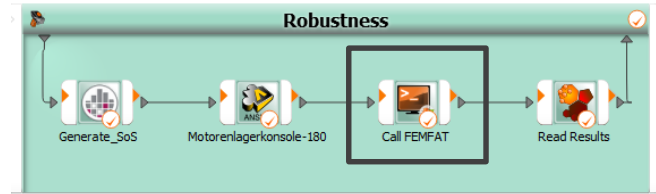
**HEAT** Sehittoglu

**SPECTRAL**

**SPOT** Remeshing

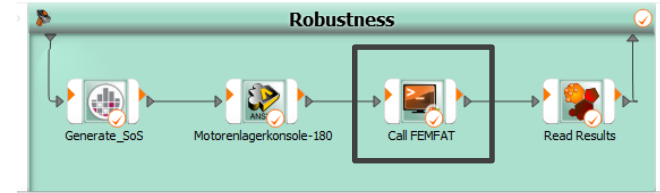
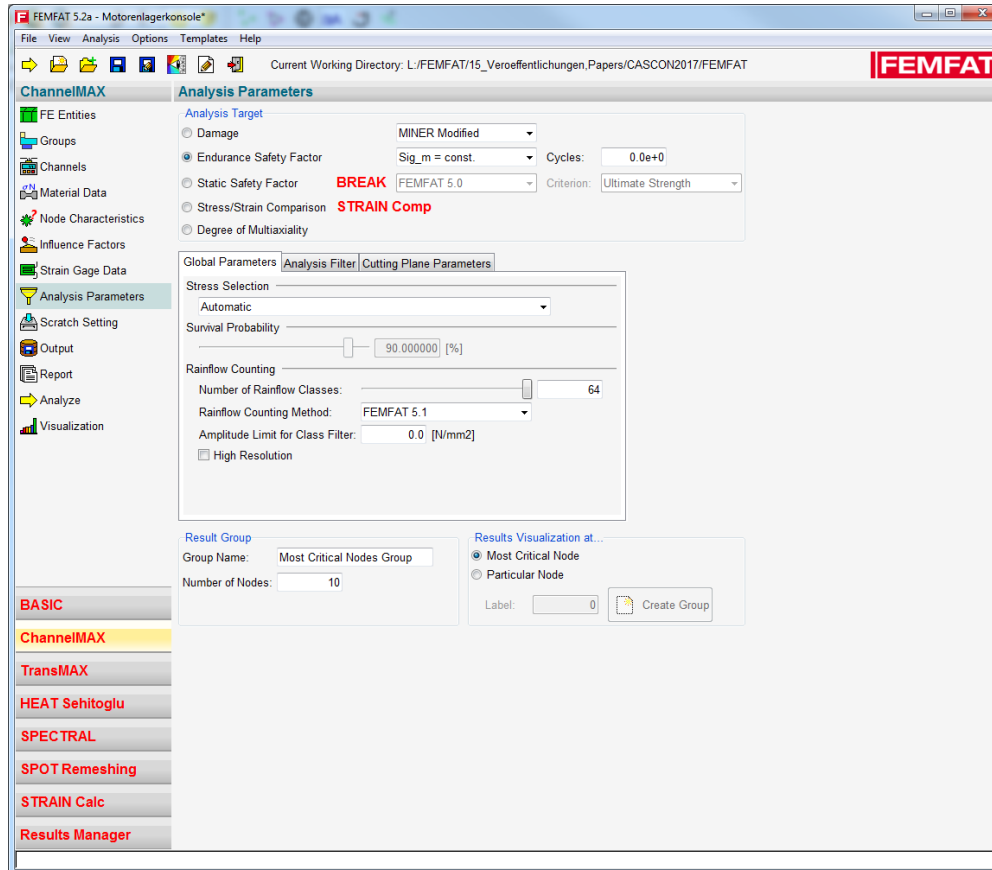
**STRAIN** Calc

**Results Manager**





# The analysis target is an „Endurance Safety Factor“ with respect to a constant mean stress.



# The fatigue simulation is done using FEMFAT channelMax. Variations are applied in the workflow by modifying the files.

## Input

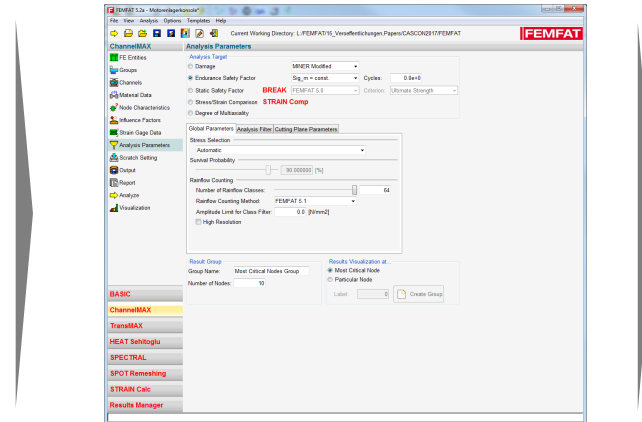
ANSYS .cdb file

ANSYS .rst file

Material file .ffd

Load history .txt

**FEMFAT** software  
FINITE ELEMENT METHOD FATIGUE



**FEMFAT** max

Stored in a .ffj file

## Output

Result file .fps

Report file .pro

# Read Results for Robustness Evaluation

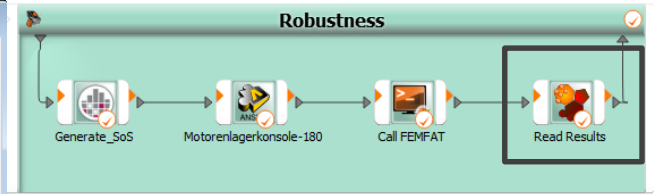
# Workflow is managed by optiSLang and the Extraction Toolkit

The screenshot shows the ETK interface with the following details:

- Title Bar:** Motorenlagerkonsole.pro - D:\Daten\AKQUISE\CASCON17\_Vortrag\Berechnung - Read Results - ETK
- Navigation:** Relative to working dir: \ AKQUISE \ CASCON17\_Vortrag \ Berechnung \ Motorenlagerkonsole.pro
- Main Text Area:**

```
7114 .
7115 .-----END OF ANALYSIS-----
7116 .
7117 .
7118 .-----MIN . ENDURANCE SAFETY FACTOR-----
7119 .
7120 .
7121 .Min. Endurance Safety Factor . . . . .0.691
7122 .in Node . . . . .58158
7123 .with Material . . . . .1 | AIS19Cu3 (Fe) F Alumi
7124 .Component . . . . .1 | Basic Material
7125 .
7126 .-----CALCULATED S/N-CURVE-----
```
- Responses Panel:** MIN\_ENDURANCE\_SAFETY\_FACTOR
- Output slots:** Standard slots
- Variable Name:** Motorenlagerkonsole
- Options:**  Instant visualization, Use as response
- Log Table:**

Date	Time	Log level	Message
------	------	-----------	---------
- Buttons:** Show additional options, OK, Cancel, Apply



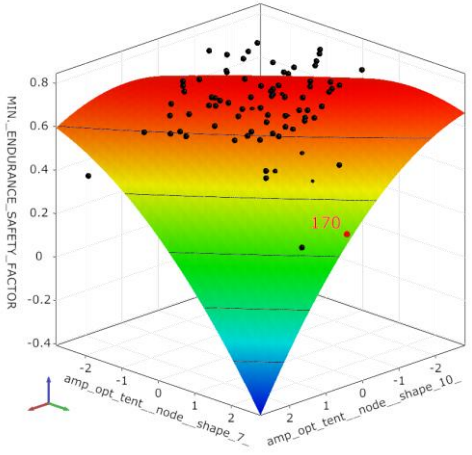
# Results

# Robustness Evaluation

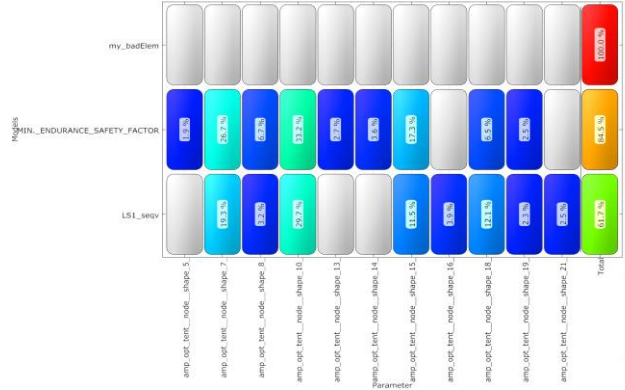
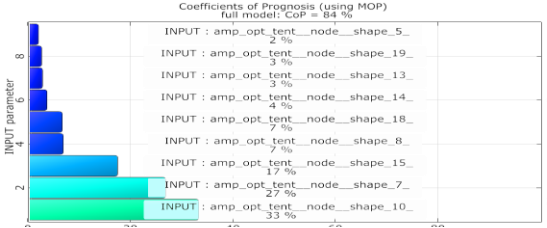
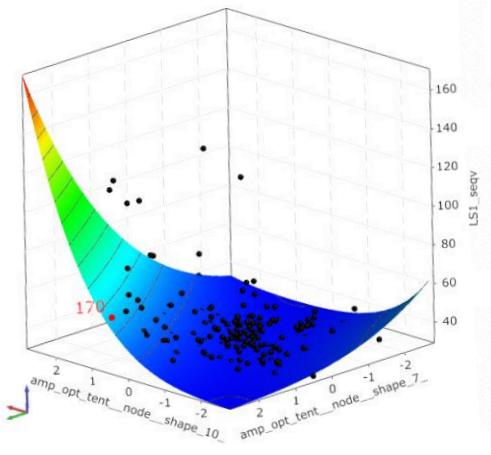


- Coefficient of Prognosis (CoP) and Metamodel of optimal Prognosis (MOP) for “Endurance Safety Factor” : 84%
- CoP for maximum stress: only 62% (due to varying location?)

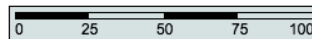
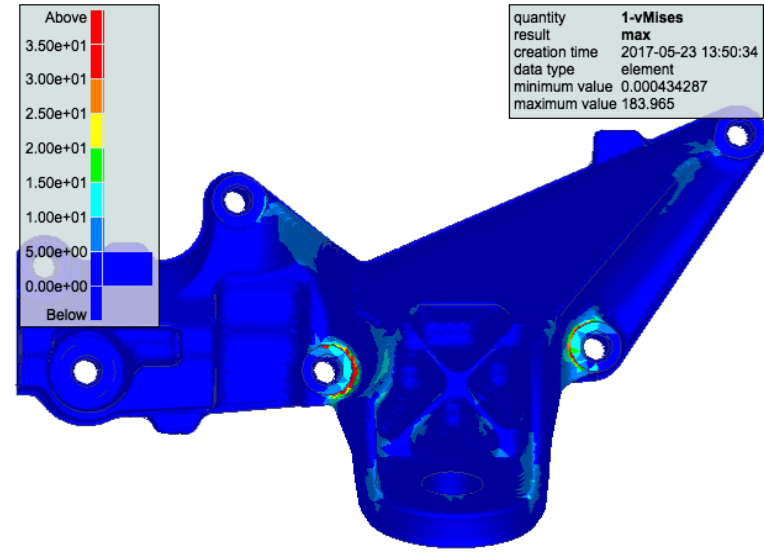
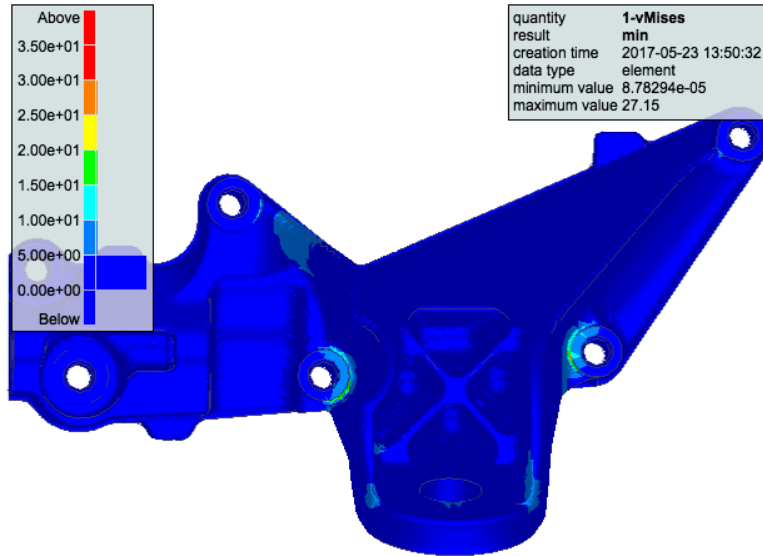
Linear Regression approximation of MIN\_ENDURANCE\_SAFETY\_FACTOR  
Coefficient of Prognosis = 84%



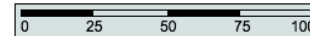
Linear Regression approximation of LS1\_seqv  
Coefficient of Prognosis = 62%



- Analysis of variation of von Mises stress due to geometric imperfection
- Left: Minimum value (safe), Right: Maximum value over all designs- unsafe

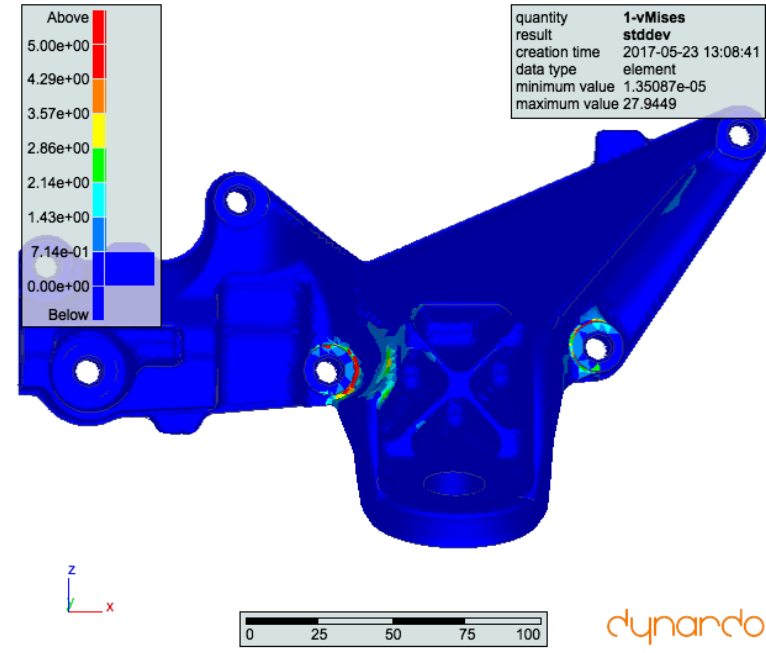
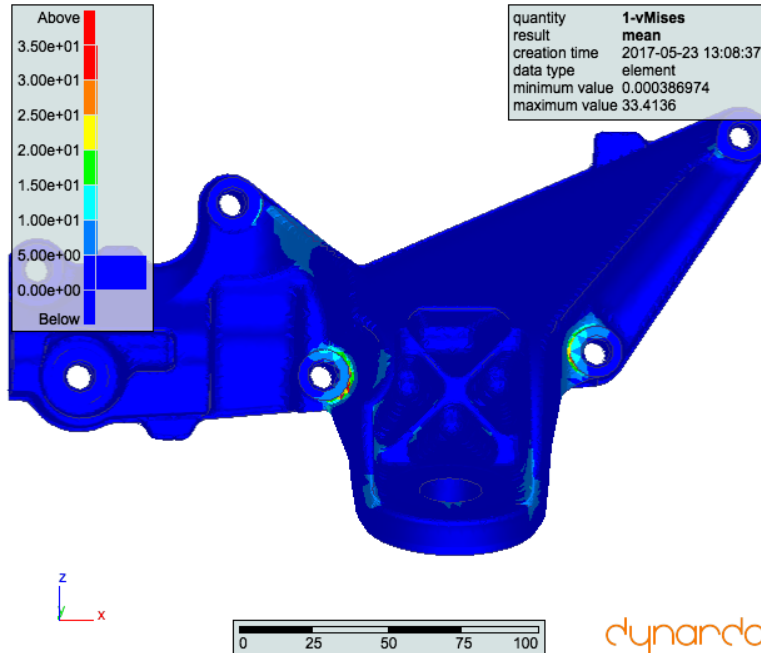


dynardo



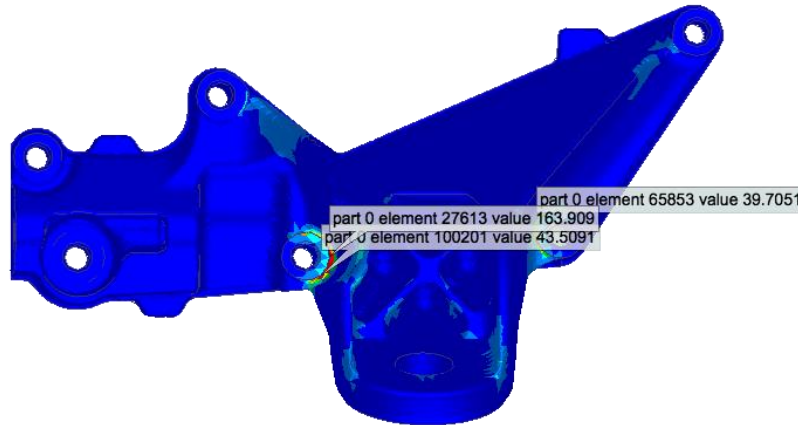
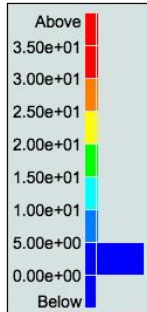
dynardo

- Analysis of variation of von Mises stress due to geometric imperfection
- Expected statistical mean value (left) and standard deviation (right)

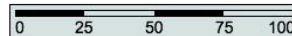




- Compute e.g. 99% quantile value, i.e. the v. Mises stress value which is exceeded by only 1% probability
- Detect “hot spots”, i.e. 3 potential failure locations exceeding the critical value

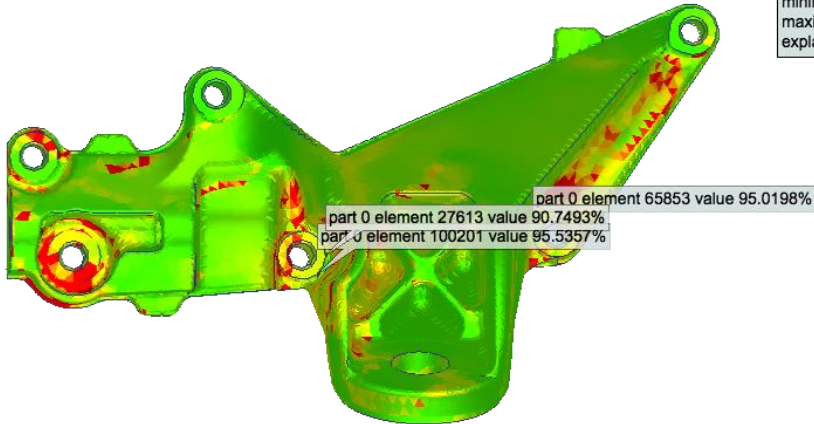
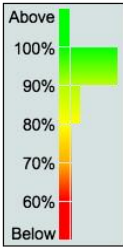


quantity	1-vMises
result	quantile[0.99]
creation time	2017-05-23 13:09:32
data type	element
minimum value	0.000427012
maximum value	163.909

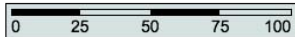


# Robustness Evaluation

- Check explainability of FMOP (field meta model):
  - 90% on average (compared with 62% before)
  - 95% at hot spots

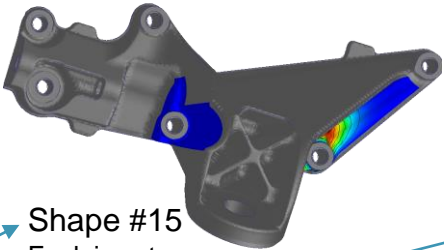
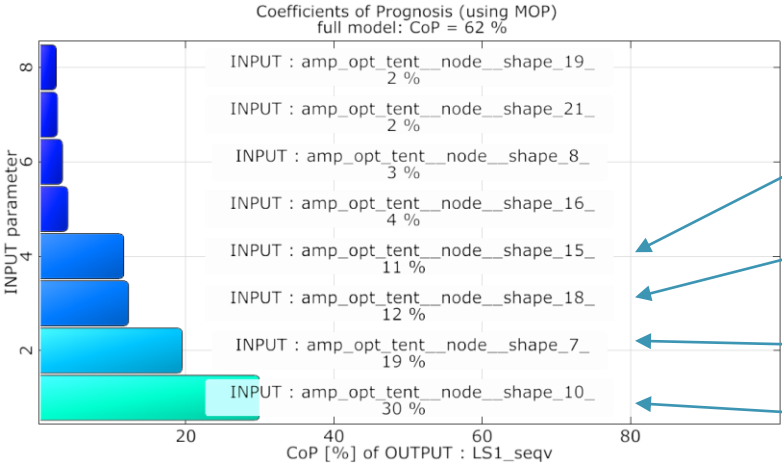


quantity	1-vMises
result	F-CoP[Total]
creation time	2017-05-23 14:30:52
data type	element
minimum value	19.1384%
maximum value	98.689%
explained variation	90.7763%

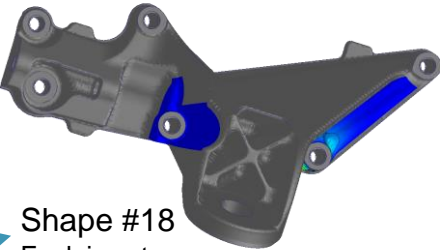


- Which geometric shape is responsible for variations ?

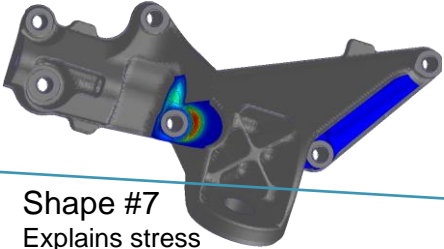
CoP matrix for maximum stress:  
(only 62% CoP)



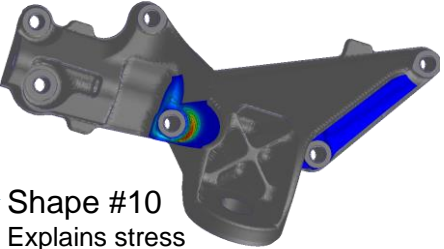
**Shape #15**  
Explains stress variation on the right hot spot by 42%



**Shape #18**  
Explains stress variation on the right hot spot by 18%



**Shape #7**  
Explains stress variation on the left hot spot by 46%



**Shape #10**  
Explains stress variation on the left hot spot by 38%

# Conclusion

# There are many applications for the use of random fields in engineering practise.

- 1 Optimisation of free form surfaces
- 2 Using statistics of measurements for geometry creation in simulations
- 3 Analysis of variations in production processes
- 4 Determination of the influence of different raw material suppliers



DRIVING **EXCELLENCE.**  
INSPIRING **INNOVATION.**