

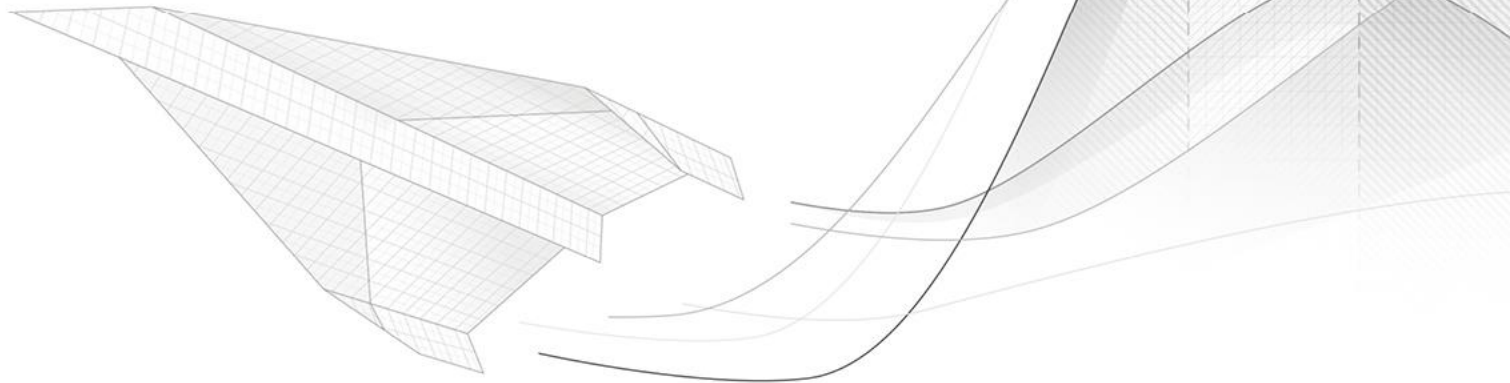


Statistics on Structures

An extension to optiSLang[®] for multi-dimensional data analysis

News & Developments

Sebastian Wolff. SoS Product Manager



Summary

Current version: SoS 7

- New features
 - ANSYS Mechanical SoS plugin
 - Interactive 3D/ROM Viewer
 - Free-form shape optimization
 - Binary interface, ANSI C (FMOPSolver.DLL)
- Improvements
 - New FMOP Backend, improved SignalMOP
 - Mesh smoothening, free-form models, named selections
 - File formats
- In Development
 - SoS Script DLL



New features: ANSYS Mechanical SoS Plugin

- Objectives:
 - Create random fields for geometric imperfections
 - Create free-form variation models for geometric imperfections
 - Import external random fields into ANSYS (e.g. obtained from measurements)
 - Expose random field parameters to ANSYS WB parameter set
 - Export result data to SoS

- Toolbar



New features: ANSYS Mechanical SoS Plugin

- Create geometric variations directly in ANSYS Mechanical
- Here: Preparation

The screenshot displays the ANSYS Mechanical interface with the following components:

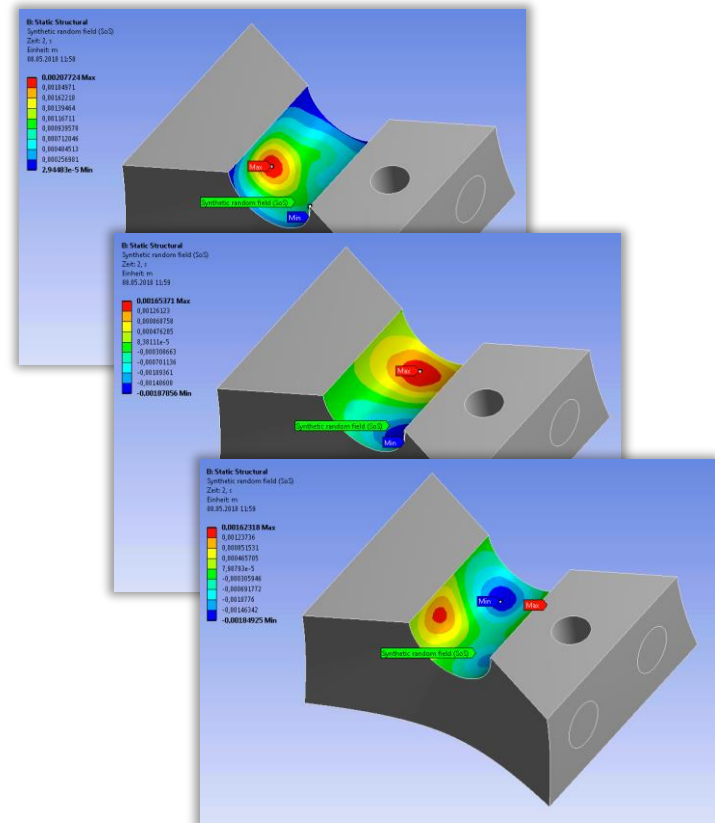
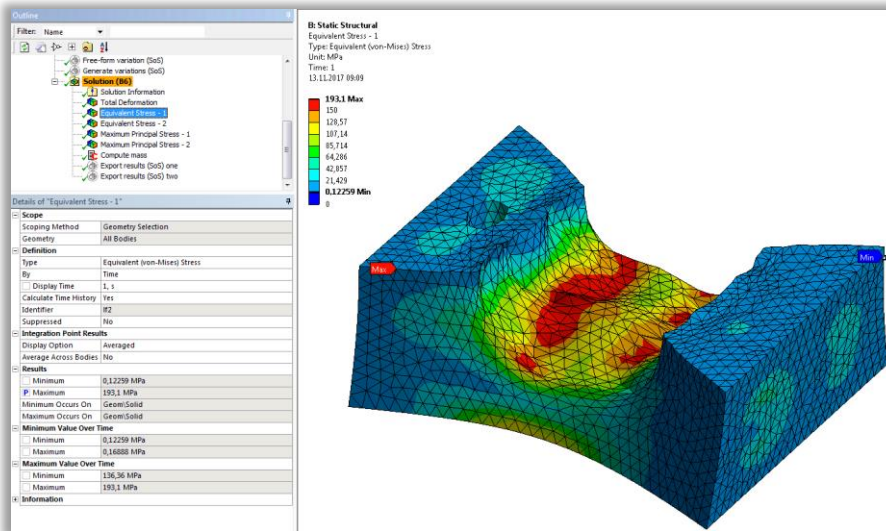
- Outline:** Shows a project structure with 'Model (B4)' containing Geometry, Coordinate Systems, Remote Points, Mesh, and Named Selections. 'Static Structural (B5)' includes Analysis Settings, Fixed Support, Remote Force, Free-form variation (SoS), and Generate variations (SoS). 'Solution (B6)' includes Solution Information, Total Deformation, Equivalent Stress, Maximum Principal Stress, Compute mass, and Export results (SoS).
- Details von "Synthetic random field (SoS)":**

Boundary to be parameterized	
Scoping Method	Named Selection
Named Selection	Boundary_to_be_morphed
Fixed boundary	
Scoping Method	Named Selection
Named Selection	Boundary_to_be_fixed
Mesh part to be exported to SoS	
Scoping Method	Named Selection
Named Selection	NS_Solid
Definition	
Desired variability (%)	90
Maximum number of parameters	10
Correlation length (eg. ~20% of total length)	0,005 [m]
Standard deviation of geometric variation	0,005 [m]
Mean geometric variation	0 [m]
Visualization	
Visible variation shape index (0=mean)	0
Advanced options	
Use mesh stabilization	True
Test on mesh distortion	False
Move nodes along	Boundary normal
Solver options	
Number of CPUs used by SoS (0=all)	0
Internal directory	SoS_SynthRF
- Details of "Generate variations (SoS)":**

SoS simulation data location	
Source item (internal)	SoS_FreeForm
Simulation data path (internal)	SoS_Generate
Solver options	
Number of CPUs used by SoS (0=all)	2
Input parameters	
SoS input ident	amp_model__node__shape_1_
P Value	-1,81609602420301
SoS input ident	amp_model__node__shape_2_
P Value	-2
SoS input ident	amp_model__node__shape_3_
P Value	0,138171074358871
SoS input ident	amp_model__node__shape_4_
P Value	-2
SoS input ident	amp_model__node__shape_5_
P Value	-2
SoS input ident	amp_model__node__shape_6_
P Value	-2
SoS input ident	amp_model__node__shape_7_
P Value	-2
SoS input ident	amp_model__node__shape_8_
P Value	-0,0905465028254047
SoS input ident	amp_model__node__shape_9_

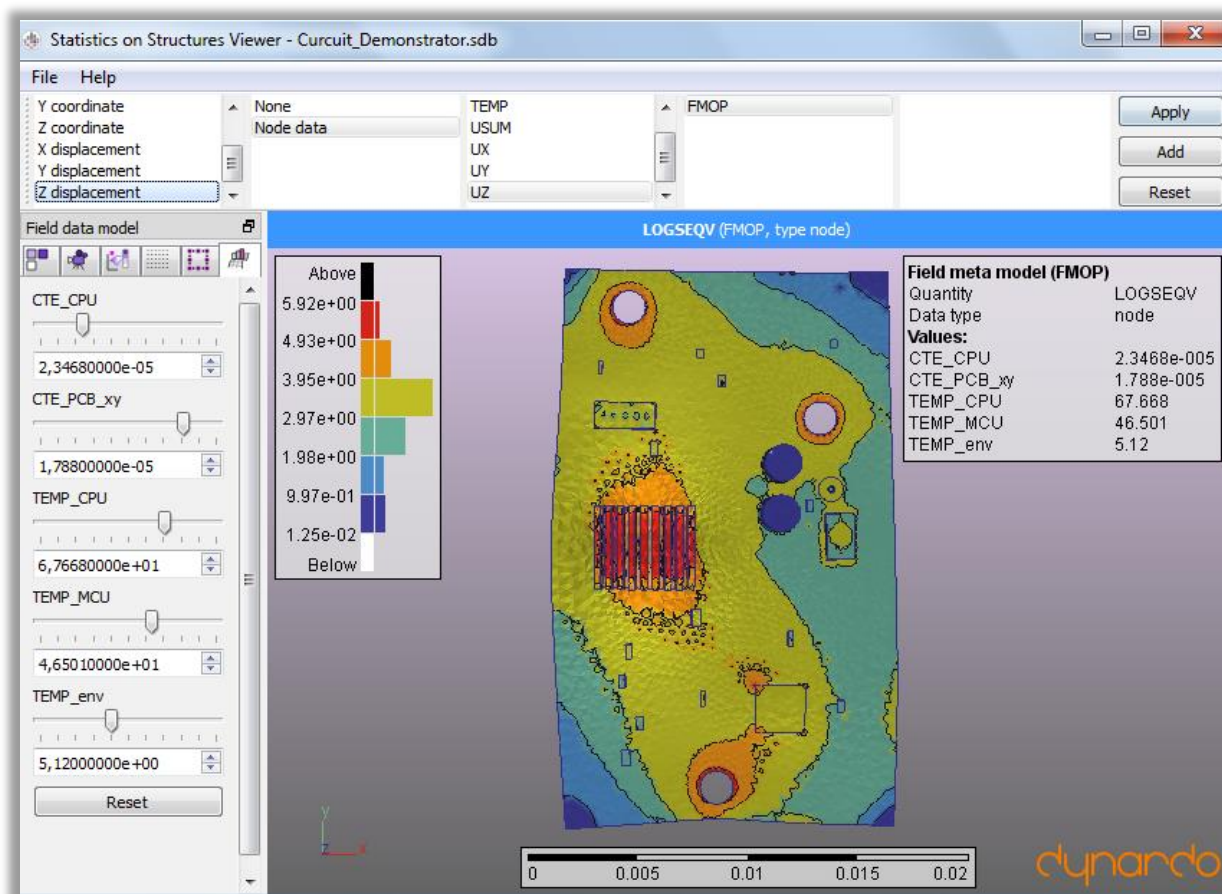
New features: ANSYS Mechanical SoS Plugin

- Visualize scatter shapes, visualize applied variation as contour plot or deformation in ANSYS Mechanical



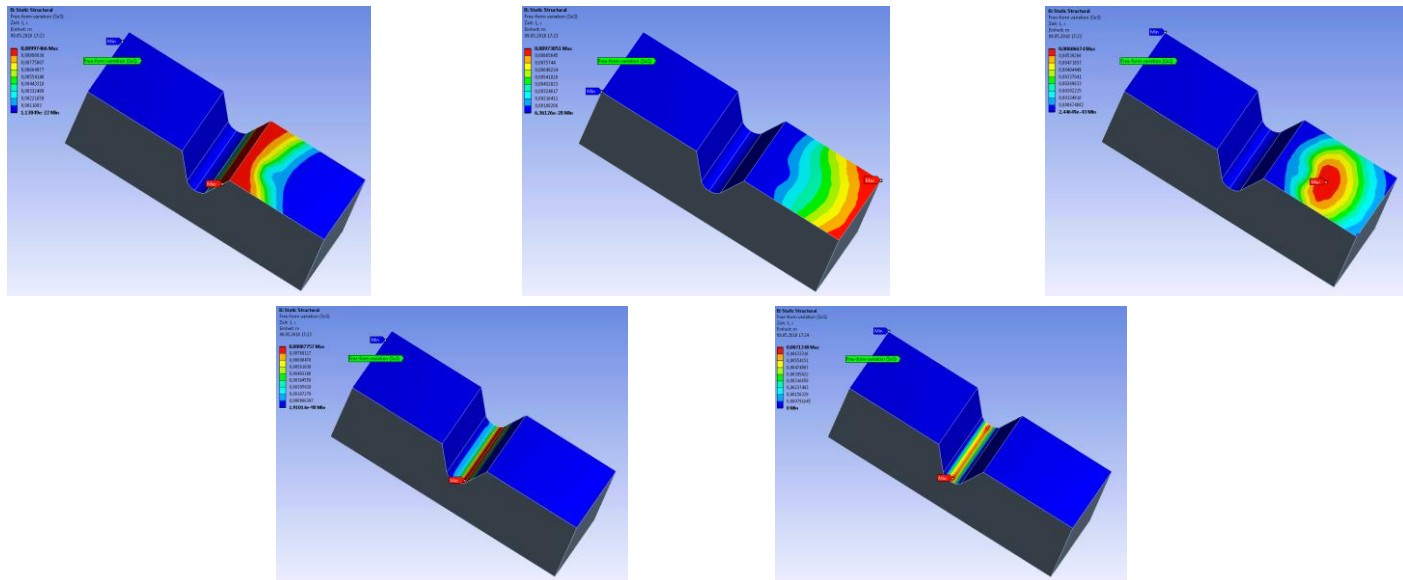
New features: Interactive 3D/ROM Viewer

- **Standalone SoS Viewer** for FMOP, Random Fields and statistical analysis (read-only mode)
- **Interactive visualization:** Use *sliders* to change parameters of FMOP and RF
- Visualize data as colored **contour plots** and **deformed geometry**



New features: Free-form shape variations

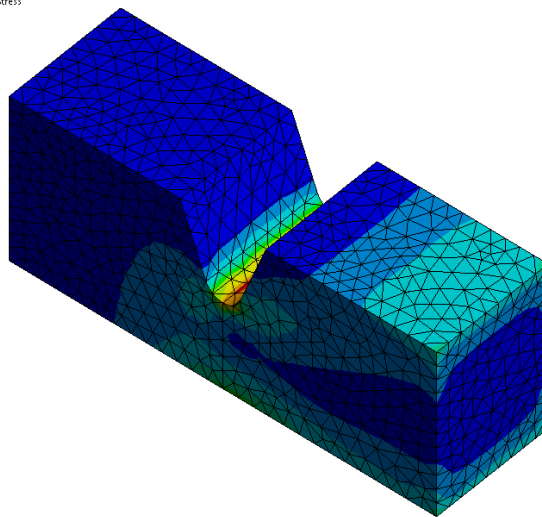
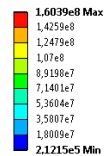
- New random field model type: Free-Form variation models
- What is new ?
 - Classical random fields: Global shapes ("sine"- "cosine"- alike)
 - Free-Form fields: Local shapes using support points
 - Support points can be placed automatically or manually or both



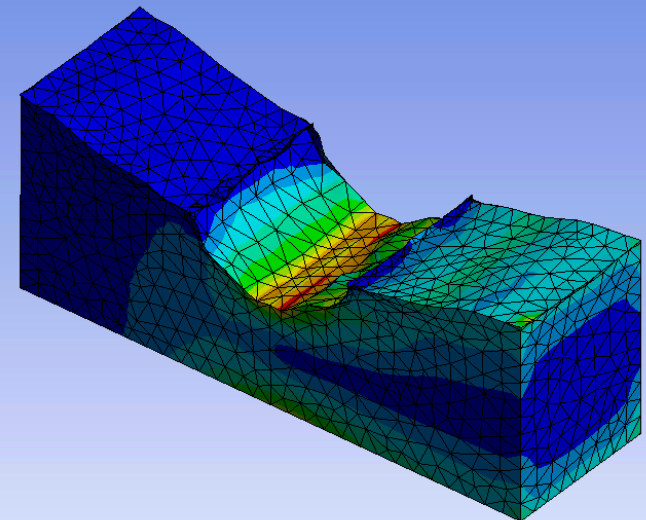
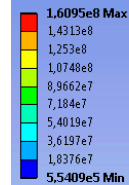
New features: Free-form shape variations

- Successful mesh morphing even for complex geometries...

B: Static Structural
Equivalent Stress 2
Type: Equivalent (von-Mises) Stress
Unit: Pa
Time: 1
09.05.2018 17:33



B: Static Structural
Equivalent Stress 2
Typ: Vergleichsspannung (von Mises)
Einheit: Pa
Zeit: 1
09.05.2018 17:30



New features: FMOPSolver.DLL

- FMOPSolver.dll / .so for Windows + Linux
- ANSI C API for
 - Dynamic linking with C/C++ software
 - Bindings to Python
 - Bindings to Matlab
 - Etc.
- Allows approximation of
 - signals and
 - FEM solutions (node data / element data) through FMOP

```

1 param_ids = ctypes.POINTER( ctypes.c_char_p )()
2 num_ids = ctypes.c_ulonglong(0)
3 sos.FMOP_getModelParamIds ( fmop, ctypes.byref( param_ids ), ctypes.byref ( num_ids ) )
4
5 num_mesh_items = ctypes.c_ulonglong(0)
6 sos.FMOP_getModelDim ( fmop, ctypes.byref ( num_mesh_items ) )
7
8 param_values = ( ctypes.c_double * num_ids ) ( 1., 2., 3., 4., 5., 6. )
9 approx_field = ( ctypes.c_double * num_mesh_items.value ) ( )
10 sos.FMOP_approxField ( fmop, param_values, ctypes.byref( approx_field ) )

```

ANSI C/C++

```

305
326 DYNARDO_FMOP_API fmop_error_t FMOP_getModelIds
327 ( const fmop_db_handle_t database, fmop_dataobjec
349 DYNARDO_FMOP_API fmop_error_t FMOP_getModelParamIdent
350 ( const fmop_handle_t fmop, char *** const param
370 DYNARDO_FMOP_API fmop_error_t FMOP_getParamLowerBound
390 DYNARDO_FMOP_API fmop_error_t FMOP_getParamUpperBound
407 DYNARDO_FMOP_API fmop_error_t FMOP_getModelTotalAvgFC
426 DYNARDO_FMOP_API fmop_error_t FMOP_getModelAvgFCoP (
443 DYNARDO_FMOP_API fmop_error_t FMOP_getModelDim ( cons
469 DYNARDO_FMOP_API fmop_error_t FMOP_getDataPointIndize
499 DYNARDO_FMOP_API fmop_error_t FMOP_getDataPointCoors
500
502
503 /*****/
509
535 DYNARDO_FMOP_API fmop_error_t FMOP_approxField
536 ( const fmop_handle_t fmop, const double * param
537

```

Python example

In development: SoS script DLL

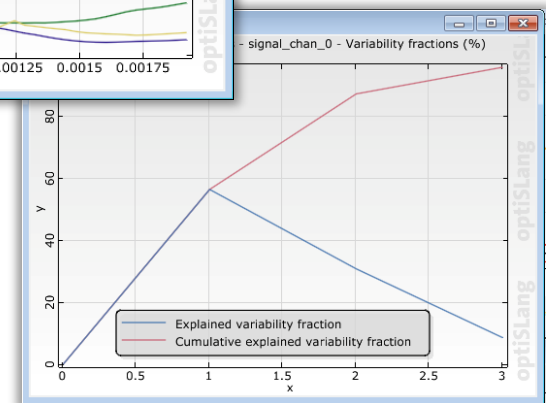
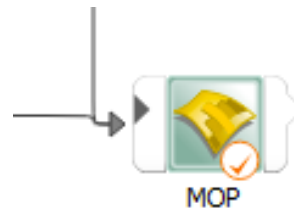
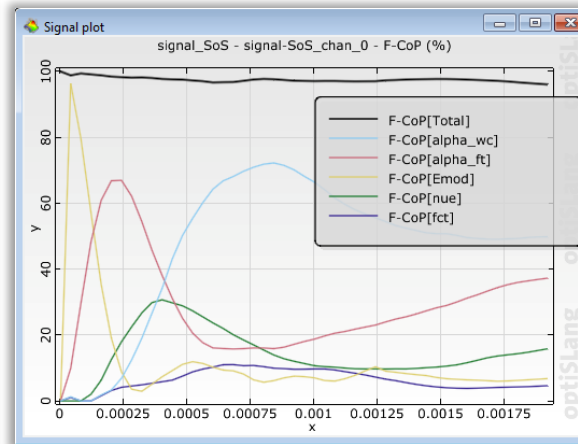
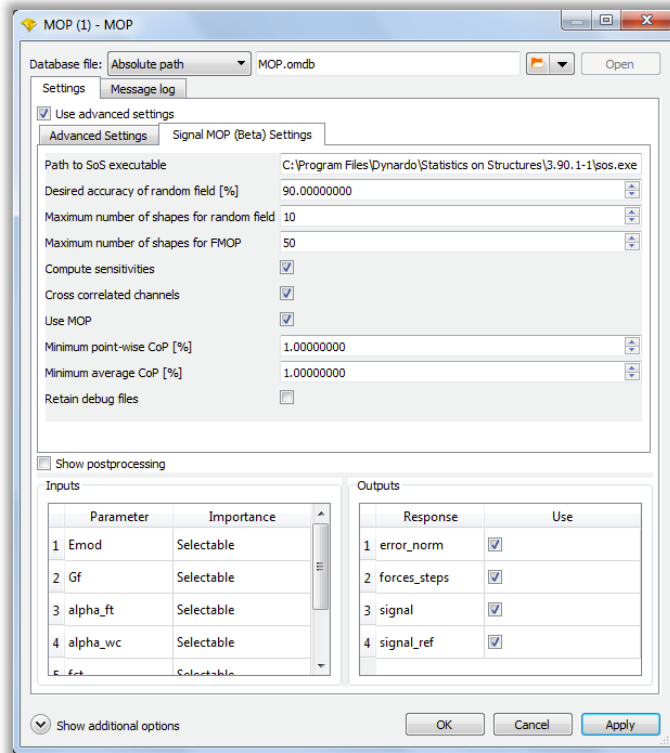
- Under development
 - Access all functions of SoS using the SoS script language
 - Start SoS scripts (internally as strings)
 - Access data through binary access
- ANSI C API (also usable for embedding into Python, Matlab etc.)

Improvements: New FMOP backend

- New meta modelling technologies in SoS
- Currently used to compute sensitivity indices and prognosis quality
 - New: Also exact for arbitrary nonlinear models
- Beta option (SoS script or SignalMOP):
 - Also used for approximation
- **FMOP can nearly reproduce same results as point-wise MOP !**

Improvements: New SignalMOP (Beta option in optiSLang 7.1)

- SignalMOP integrated into MOP and MOP-Solver nodes (using SoS 3.3.4 + 7)
- New post processing options: F-CoP, Scatter shapes, quantile values, variability fractions, linear correlations



Improvements: Support for Named Selections / Components

- Components = Named Selections, Node sets, Element sets
- Used for:
 - Import data only for a subset of your FEM mesh
 - Define fixed supports when modifying geometries

The screenshot displays the 'Set node/element reference set' dialog box in the foreground. The dialog has three input fields: 'Node set used for statistics/models', 'Fixed node set', and 'Element set used for statistics/models'. A dropdown menu is open, showing the following options: 'All nodes', 'SOS_FIXED_NODES', 'SOS_MODEL_NODES', and 'All elements'. The 'All nodes' option is currently selected. Below the dropdown are 'OK' and 'Cancel' buttons.

In the background, the main software window shows a 3D model of a mechanical part with a random field visualization. The visualization is a color map showing stress or displacement distribution. A 'Random field Results' panel is visible on the right, showing the following data:

Quantity	test2
Data type	node
Values:	
amp[test2][node_shape[1]	2.3
amp[test2][node_shape[2]	-1.31
amp[test2][node_shape[3]	0.98

At the bottom of the screenshot, a log window is visible with the following messages:

```

print("Add your own SoS script code here");
Log messages
Message
Finished: Set new reference mesh
  Set new reference node set.
  Create a free-form variation model.
  
```

The log window also shows a table with columns: Level, Thread, Time, Date.

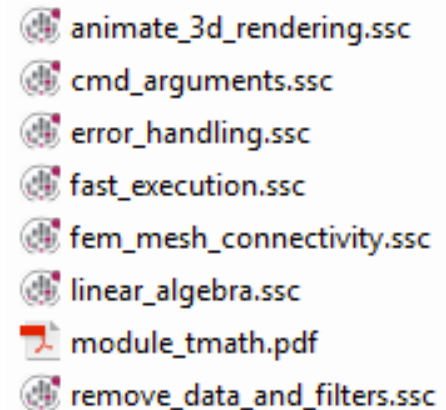
Level	Thread	Time	Date
INFO	0	17:05:16	2018-06-18
INFO	0	17:06:03	2018-06-18
WARNING	0	17:06:32	2018-06-18

Improvements: File formats and Import mappers

- Photographic measurements:
 - Import images of different resolution (resample/scale or cut-off)
- ANSYS:
 - Import named selections from CDB
 - Export data to External Data CSV format (as input to Mechanical)
- Abaqus:
 - Extend FEM element library (new: quadratic volume elements)

Improvements: Advanced SoS script API

- Powerful and fast SoS script API for automated processing
- See examples (C:\Users\Public\Documents\Dynardo\...)
- Used within projects e.g. to
 - Do simple fatigue analysis based on FMOP stress results (implementing rainfall counting)
 - Create a database for worn brake pad measurements
 - Perform time stepping based on nonlinear ROM (through FMOP)
 - Much more....



Script examples in
C:\Users\Public\Documents



dynardo

dynamic software & engineering

contact@dynardo.at
Dr. Sebastian Wolff