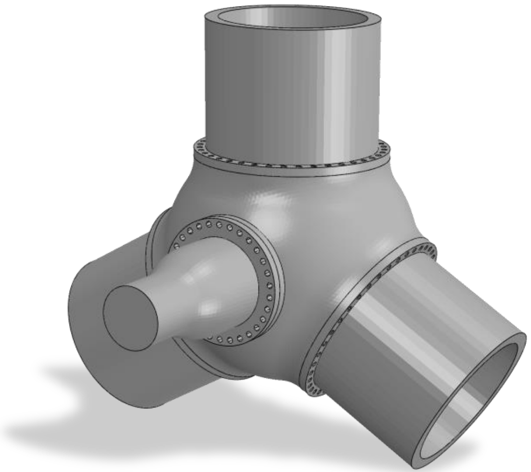
The Tosca Tuesdays

Tosca Tuesday #5

Basics: Shape optimization

Example: Shape optimization of a wind turbine hub



Basics | Shape optimization

TTT
#4

Fundamental concept

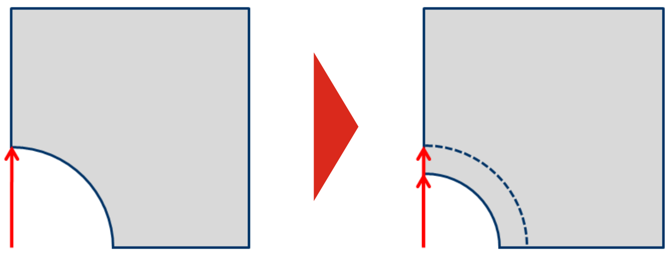
- **Design variables:** Position of each surface node from a given design area
- **Goal:** Calculate an optimal surface geometry of a given model under consideration of all boundary conditions, constraints and geometric restrictions
- **Result:** Optimization displacements for all corresponding design nodes
- **Examples of possible tasks (controller strategy):**
    - Minimize local stress /strain peaks
    - Minimize local stress / strain peaks under a volume equality constraint
    - Minimize local fatigue hotspots / nodal damage peaks
    - Minimize local fatigue hotspots / nodal damage peaks under a volume equality constraint
    - …

Basics | Shape optimization

TTT #4

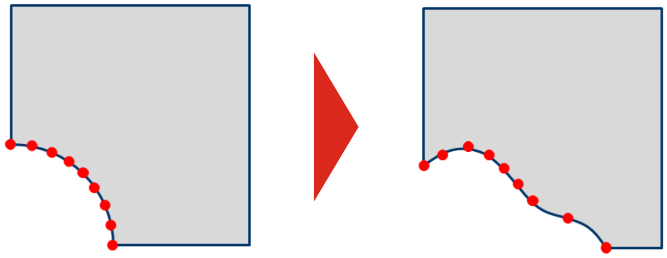
Different approaches

Parametric concept (→ Isight)



e.g. Variation of a radius

Non-parametric concept (→ Tosca)

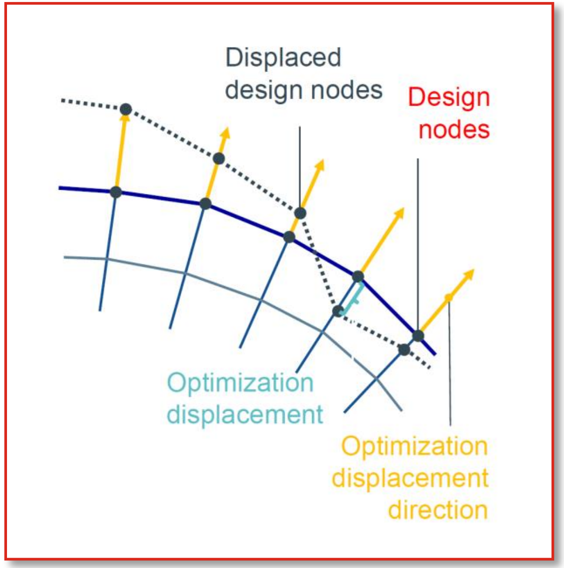


Modification of each surface node

→ Flexibility of possible shapes

Basics | Shape optimization

Non-parametric concept



Terminology

- **"Growth":** Design node is moved outwards in normal direction (positive displacement)
- **"Shrinkage":** Design node is moved against normal direction (negative displacement)
- The **design variables** are more specifically the optimization displacement values of the corresponding design nodes.

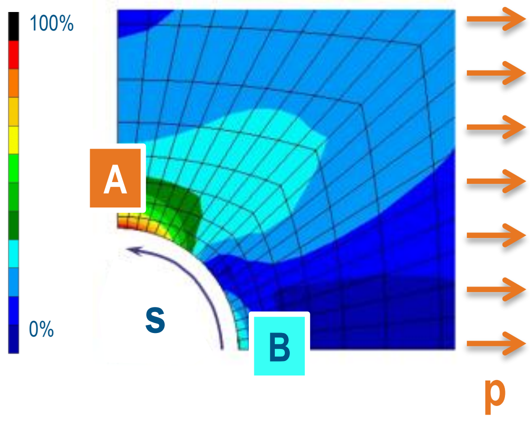
Basics | Shape optimization

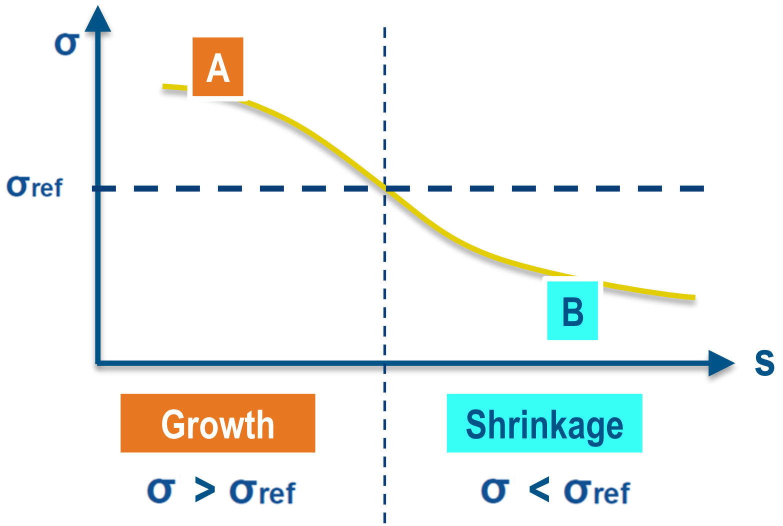
Controller strategy

- **Goal:** Minimize local stress peaks on the surface of a structure
- **Controller strategy:**
  - Homogenization of the surface stress distribution
  - “Growth” in regions of high stress → leads to local stress decrease
  - “Shrinkage” in regions of low stress → leads to local stress increase
  - Characterization of a certain region by using a reference stress (~ average stress of all design nodes)
- **Result:** Homogeneous (uniform) stress distribution without any stress peaks
- The concept is in a similar manner applicable to strain or damage peaks (fatigue analysis).

Basics | Shape optimization

Controller strategy – Example of a hole plate





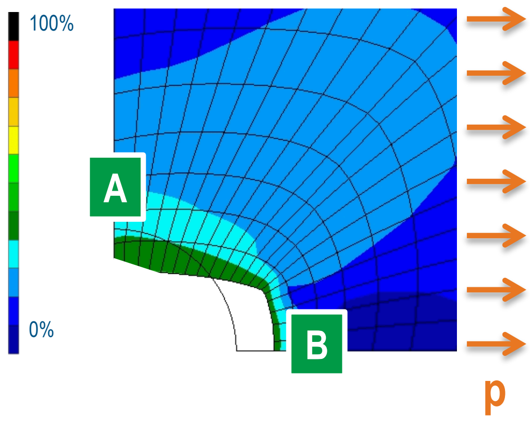
SIMULIA

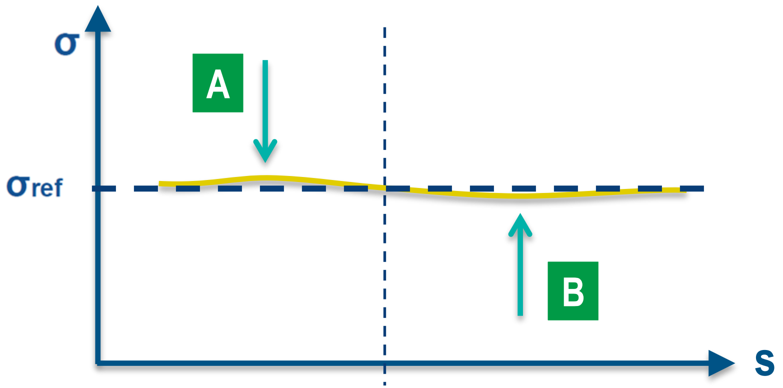
DASSAULT SYSTEMES | The 3DEXPERIENCE® Company

Basics | Shape optimization

TTT
#4

Controller strategy – Example of a hole plate



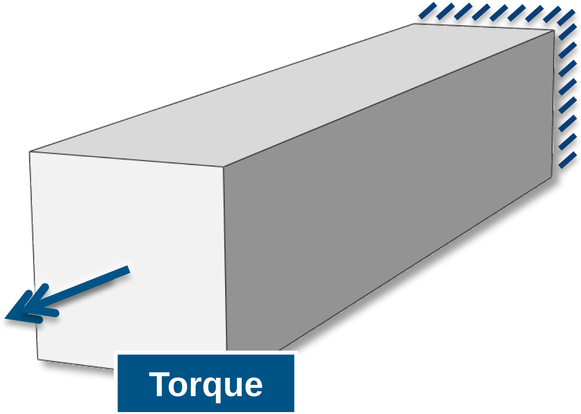


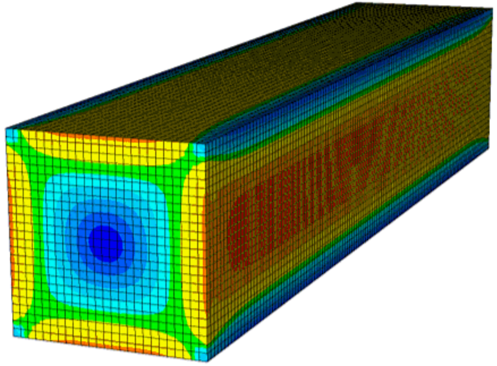
Homogenous stress distribution

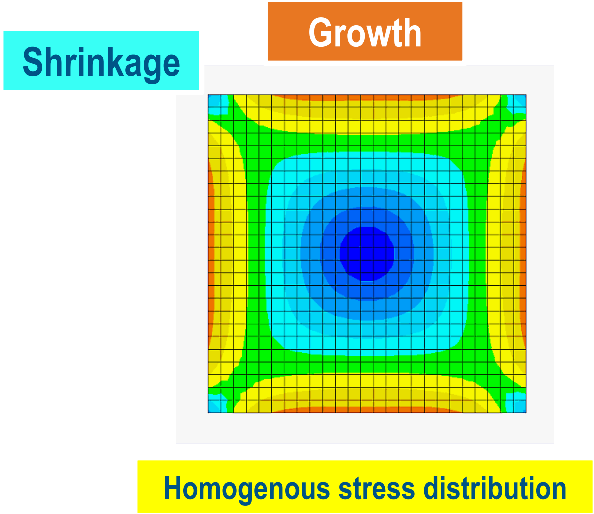
Basics | Shape optimization

TTT #4

Controller strategy – Example of a rectangular beam under torsion





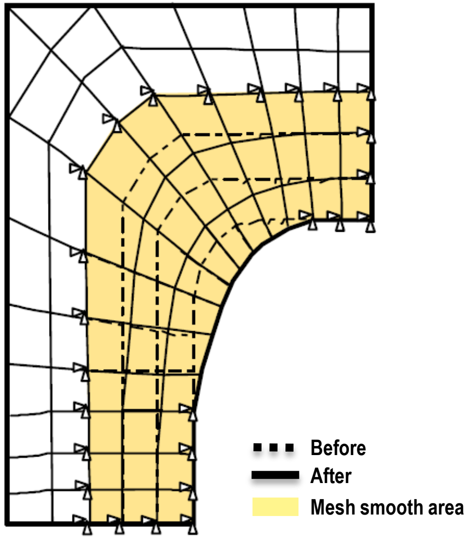


Basics | Shape optimization

NEW

Mesh smoothing

- **Modifications during shape optimization:**
  - Displacement of the surface nodes due to local stresses
  - Distortion of elements near to the surface layer
  - Quality of the finite element analysis is affected
- **Mesh smoothing:**
  - Displacements at the design nodes are passed to the inner nodes which lie in a so-called mesh smooth area
  - Mesh adjustment and mesh smoothing takes place in each optimization iteration
  - Mesh smoothing preserves a good mesh quality



Tosca Tuesday #5

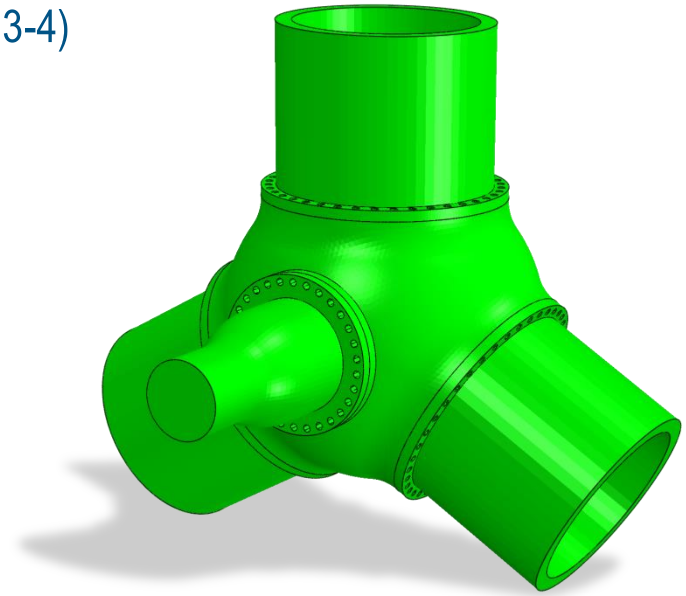
Basics: Shape optimization

Example: Shape optimization of a wind turbine hub

Example | Wind turbine hub

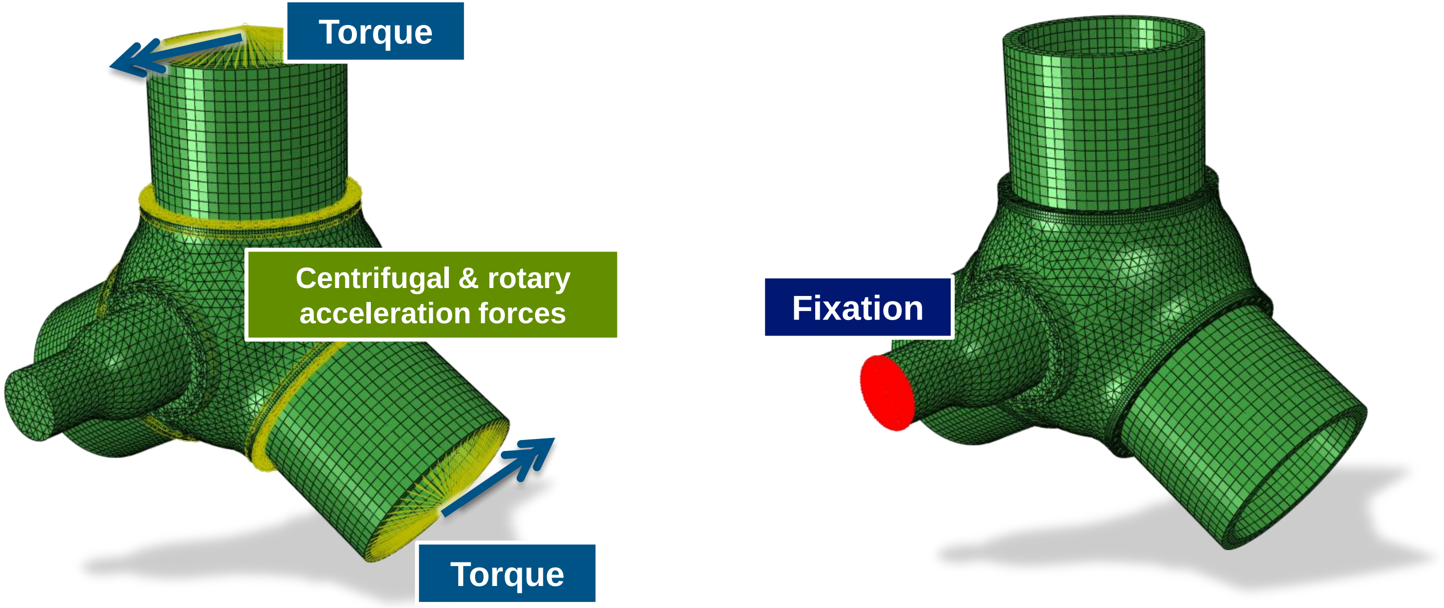
Get started …

- Start Abaqus CAE (at least version 6.13, preferable 6.13-4)
- File → Import → Model → "hub.inp"
- File → Set Work Directory → Choose Directory



Example | Wind turbine hub

Basic model: Loading and boundary conditions

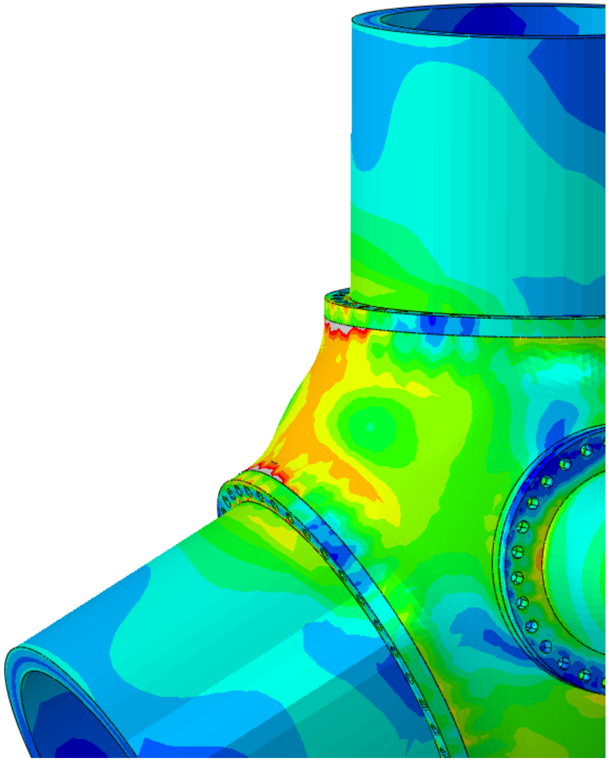


Example | Wind turbine hub

Shape Optimization: Setup

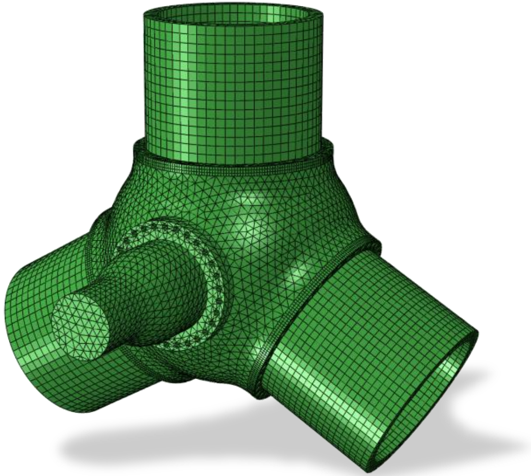
- **Objective function**
  - Minimize maximal stress peaks (v. Mises)

  → Controller strategy
- **Geometric restrictions**
  - Rotational symmetry
  - Planar symmetry for each segment

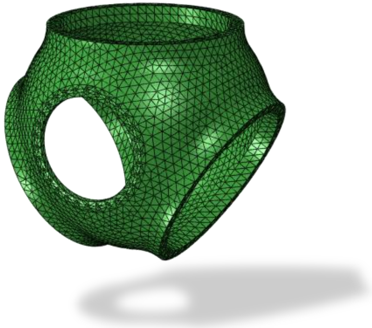


Example | Wind turbine hub

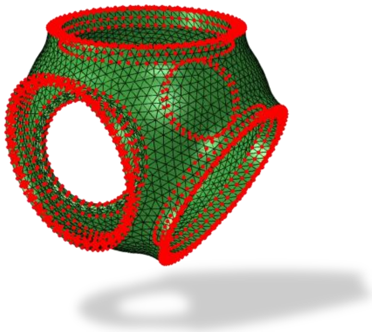
Shape Optimization: Used node sets



Complete model



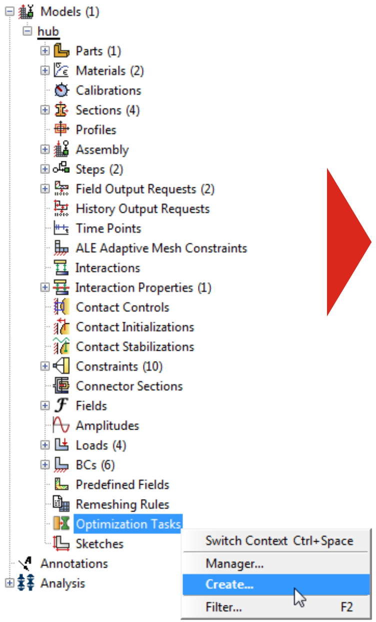
Design area

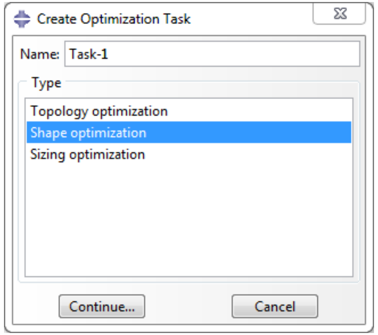


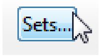
Fixed nodes

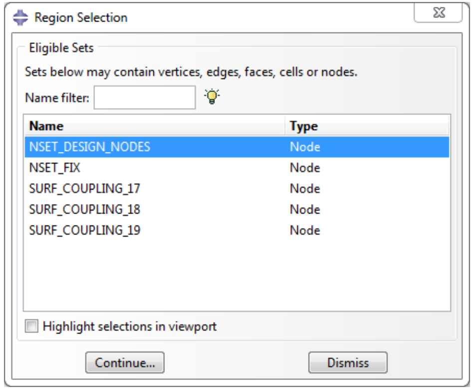
Example | Wind turbine hub

Step 1: Shape optimization task



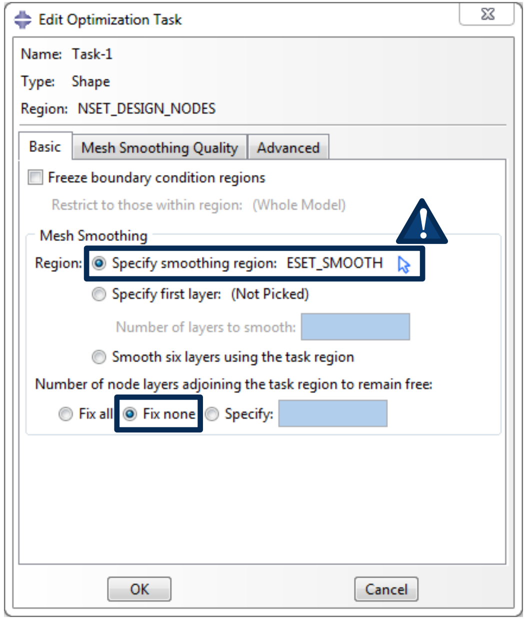


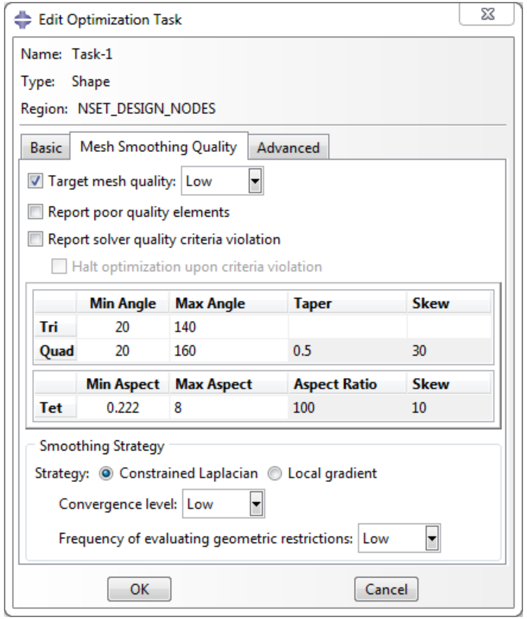


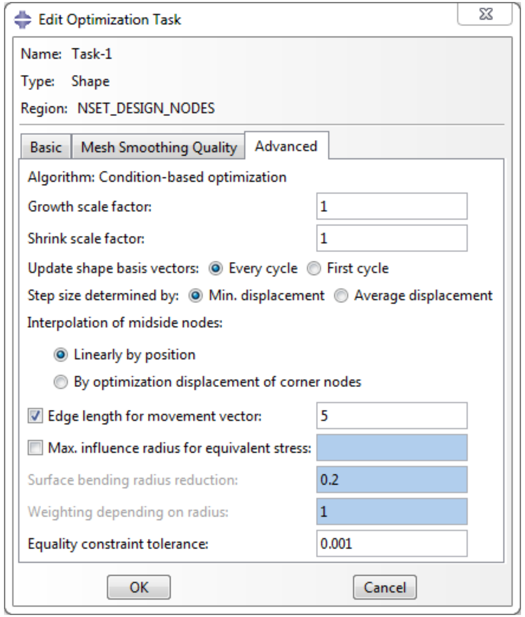


Example | Wind turbine hub

Step 1: Shape optimization task

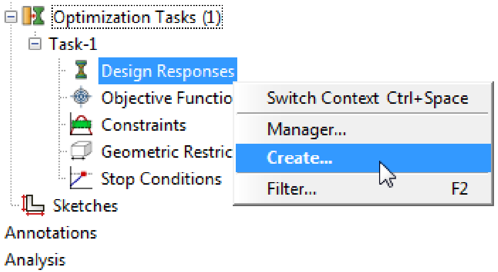


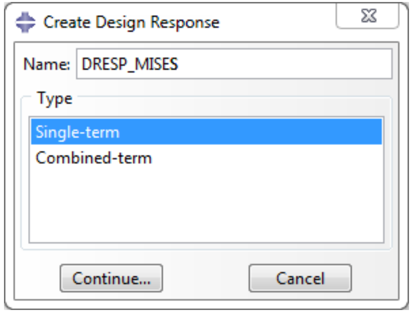


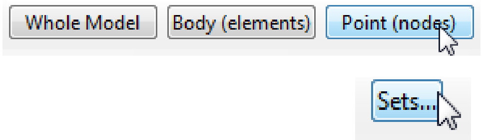


Example | Wind turbine hub

Step 2: Design response for maximal v. Mises stress

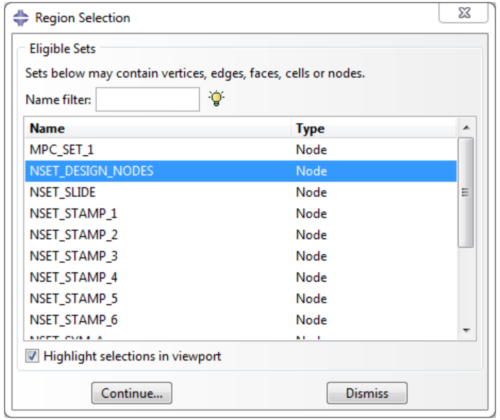


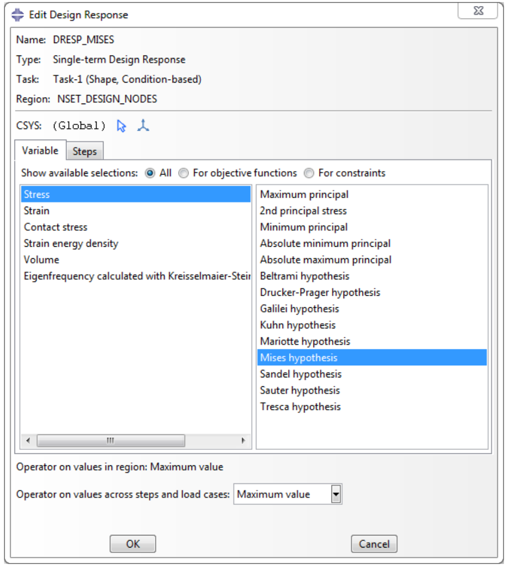


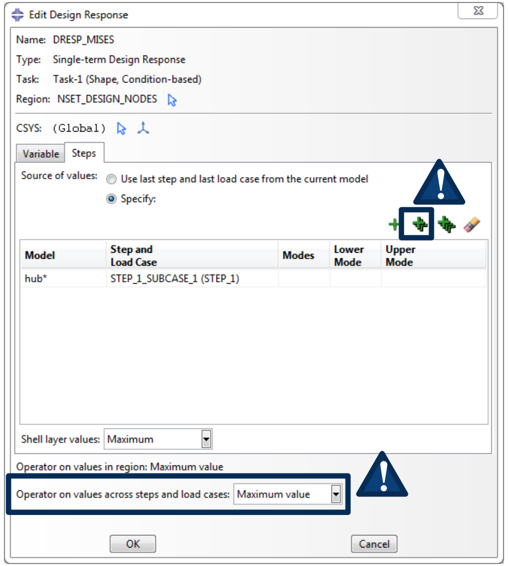


Example | Wind turbine hub

Step 2: Design response for maximal v. Mises stress



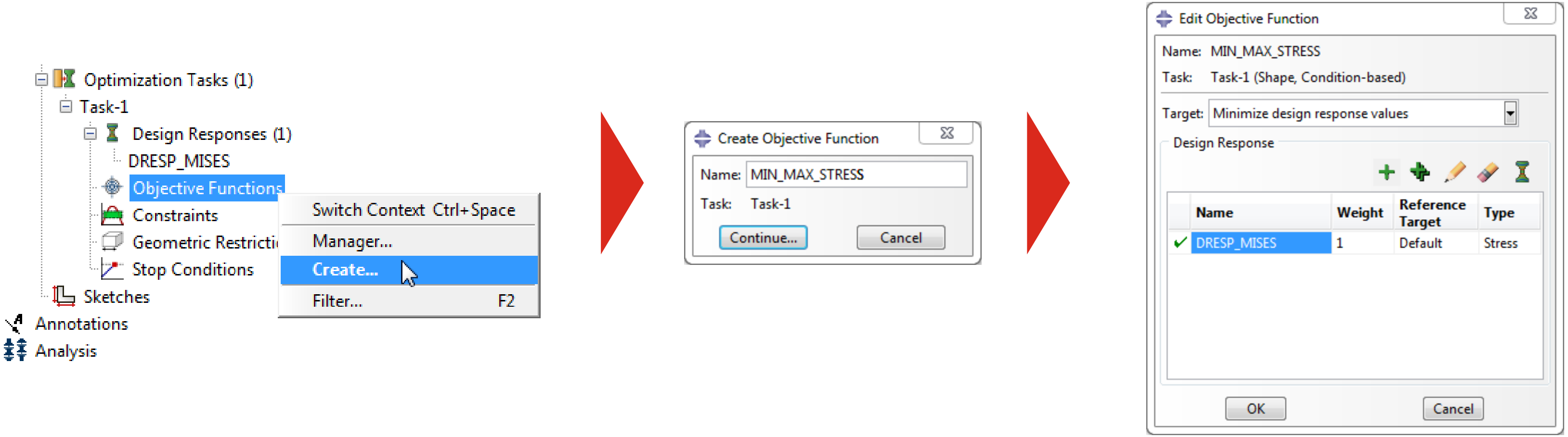




The 3DEXPERIENCE® Company

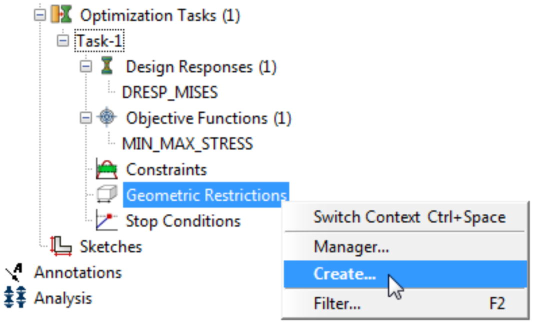
Example | Wind turbine hub

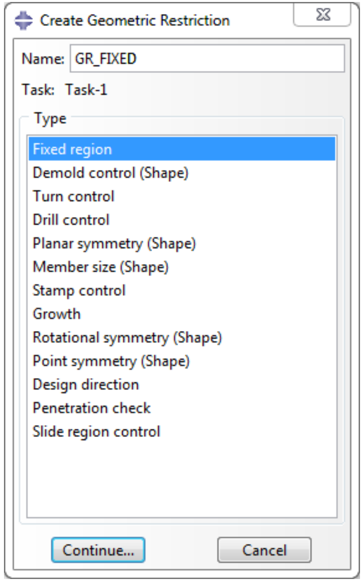
Step 3: Objective function (→ minimize maximal v. Mises stress peak)

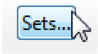


Example | Wind turbine hub

Step 4: Geometric restriction for a fixed region

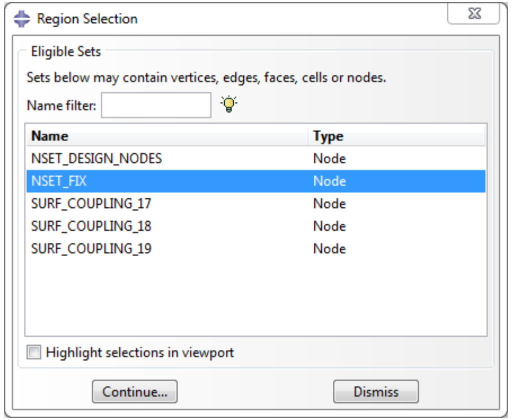


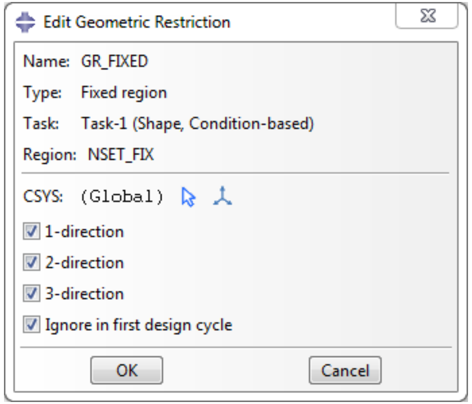




Example | Wind turbine hub

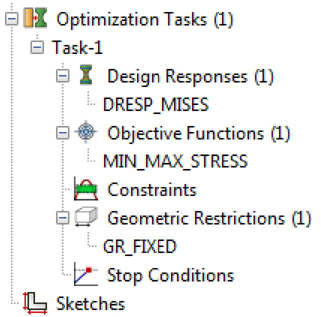
Step 4: Geometric restriction for a fixed region

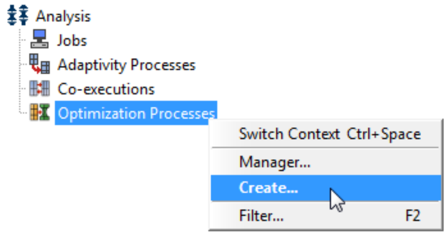


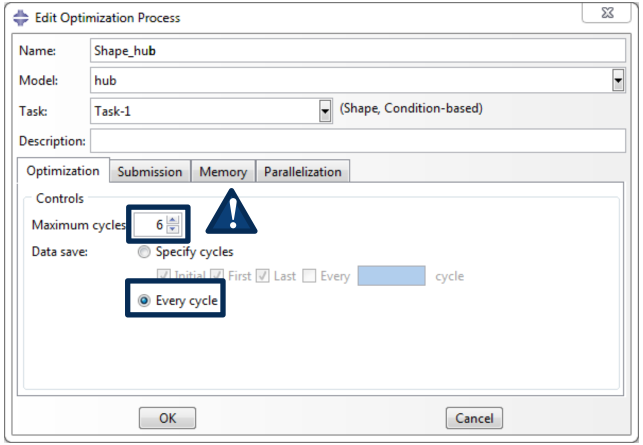


Example | Wind turbine hub

Step 5: Creation of an optimization task

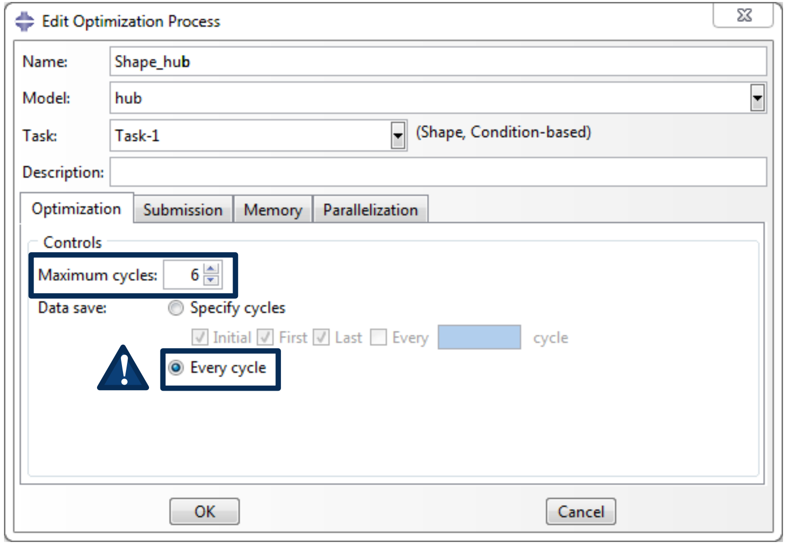


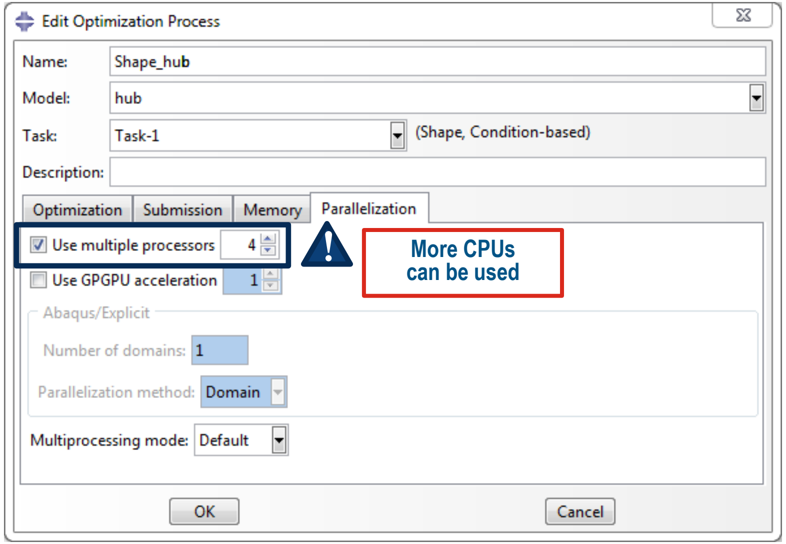




Example | Wind turbine hub

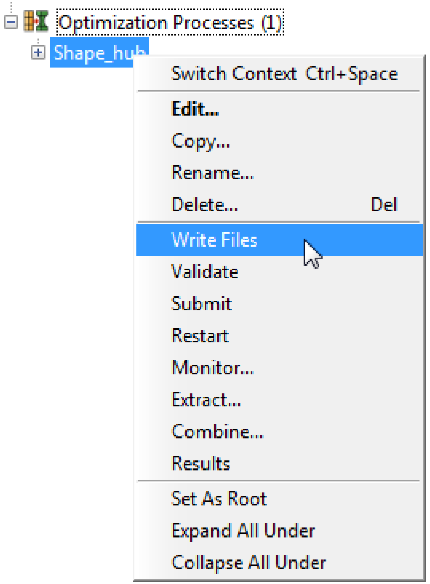
Step 5: Creation of an optimization task





Example | Wind turbine hub

Step 6: Write out the files of the optimization task



The following files are created in the defined work directory:

- Abaqus input deck:
  Shape_hub-Job.inp
- Tosca parameter file:
  Shape_hub.par

Example | Wind turbine hub

Step 7: Tosca parameter file

Open up the Tosca parameter file *“Shape_hub.par”* in a text editor

Shape_hub - Notepad

File Edit Format View Help

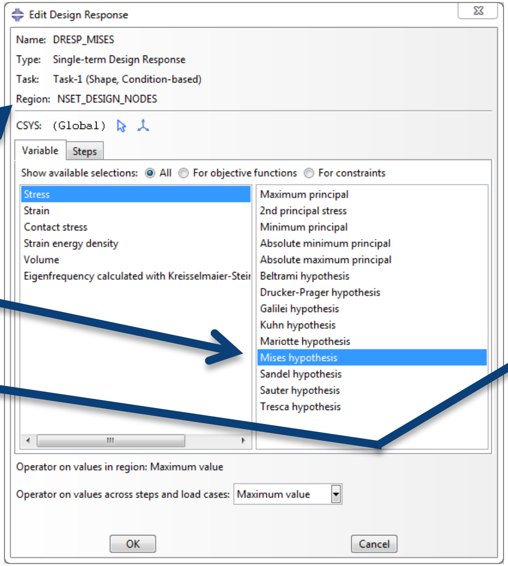
```
! Optimization Process name: Shape_hub
! Model name: hub
! Task name: Task-1
! Generated by: Abaqus/CAE 6.13-4
FEM_INPUT
  ID_NAME = Shape_hub-Job_INP_
  FILE = Shape_hub-Job.inp
END_
! ---------------------------------------------------------------
!
! Design Response: DRESP_MISES
!
DRESP
  ID_NAME = DRESP_MISES
  LIST = NO_LIST
  DEF_TYPE = SYSTEM
  TYPE = SIG_MISES
  ND_GROUP = NSET_DESIGN_NODES
  GROUP_OPER = MAX
  LC_SET = ALL, 1, ALL, MAX
  LC_SEL = MAX
END_
! ---------------------------------------------------------------
!
! Design area for task: Task-1
!
DV_SHAPE
  ID_NAME = Task-1_DESIGN_AREA_
  ND_GROUP = NSET_DESIGN_NODES
END_
MESH_SMOOTH
  ID_NAME = Task-1_MESH_SMOOTH_
  EL_GROUP = ESET_SMOOTH
  FREE_SF = FREE
  STRATEGY = CONSTRAINED_LAPLACIAN
  LEVEL_CONV = LOW
  LEVEL_DVCON = LOW
  LEVEL_QUAL = LOW
  QUAL_LIST = NO
  SOLVER_CHECK = NO
  SOLVER_STOP = NO
  TRIA_LOW_ANGLE = 20.
  TRIA_HIGH_ANGLE = 140.
  QUAD_LOW_ANGLE = 20.
  QUAD_HIGH_ANGLE = 160.
  Q4TAPER = 0.5
  Q4SKEW = 30.
  TETRA_LOW_ASPECT = 0.222
  TETRA_HIGH_ASPECT = 8.
  TETRAAR = 100.
  T3SKEW = 10.
END_
```

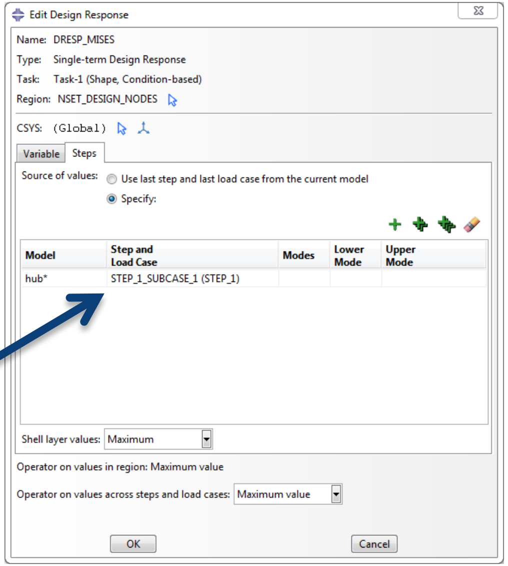
The parameter file is the fundamental settings file for Tosca. It contains all necessary data for an optimization.
(e.g. Type of optimization, design variables, objective function, constraints, restrictions…)

Example | Wind turbine hub

Step 7: Tosca parameter file – Design response

```
! -----------------------------------
!
! Design Response: DRESP_MISES
!
DRESP
  ID_NAME = DRESP_MISES
  LIST = NO_LIST
  DEF_TYPE = SYSTEM
  TYPE = SIG_MISES
  ND_GROUP = NSET_DESIGN_NODES
  GROUP_OPER = MAX
  LC_SET = ALL, 1, ALL, MAX
  LC_SEL = MAX
END_
```

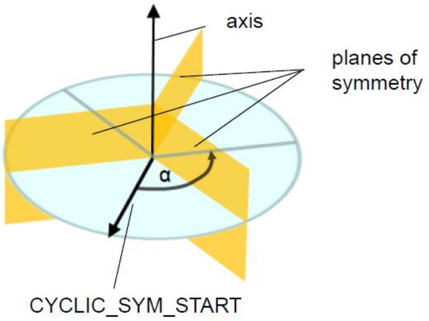




Example | Wind turbine hub

Step 8: Definition of a symmetry restriction within the parameter file

In addition to the basic optimization setup which was defined in Abaqus/CAE the following symmetry restriction is now directly added within the Tosca parameter file.



The design area is basically divided into a set of segments (in this case 3) which are repeated around a rotational axis. Additionally this constraint ensures that the optimization displacements within each segment show a planar symmetry (see yellow planes); *further information in the Tosca Structure documentation, page 2-345*

Example | Wind turbine hub

Step 8: Definition of a symmetry restriction within the parameter file

Copy the following lines into the parameter file!

```
!---------------------
!
! Link Condition: LINK_SURF_CYCLIC_PLANE_SYM
!
LINK_SHAPE
 ID_NAME = LINK_SURF_CYCLIC_PLANE_SYM
 MASTER = MAX
 CLIENT = SURF_CYCLIC_PLANE_SYM
 CYCLIC_SYM_START =  0., 0. , 1.
 CLIENT_DIR = 1.,0.,0.
 CS = CS_0
 TOL = 0.1
 ANGLE = 120.
END_
!---------------------
!
! Geometric Restrictions: GR_SURF_CYCLIC_PLANE_SYM
!
DVCON_SHAPE
 ID_NAME = GR_SURF_CYCLIC_PLANE_SYM
 ND_GROUP = NSET_DESIGN_NODES
 CHECK_BC = NO
 CHECK_LINK = LINK_SURF_CYCLIC_PLANE_SYM
 FEASIBLE_START = YES
END_
```

```
! ----------------------------------------
!
! Objective Function: MIN_MAX_STRESS
!
OBJ_FUNC
  ID_NAME = MIN_MAX_STRESS
  DRESP = DRESP_MISES, 1.
  TARGET = MIN
END_

!---- COPY BELOW THIS POSITION ------!

! ----------------------------------------
!
! Geometric Restriction: GR_FIXED
!
DVCON_SHAPE
  ID_NAME = GR_FIXED
  ND_GROUP = NSET_FIX
  CHECK_DOF = CS_0, FIX, FIX, FIX
  FEASIBLE_START = YES
END_
```

Example | Wind turbine hub

Step 8: Definition of a symmetry restriction within the parameter file

Copy the following line into the parameter file!

DVCON = GR_SURF_CYCLIC_PLANE_SYM

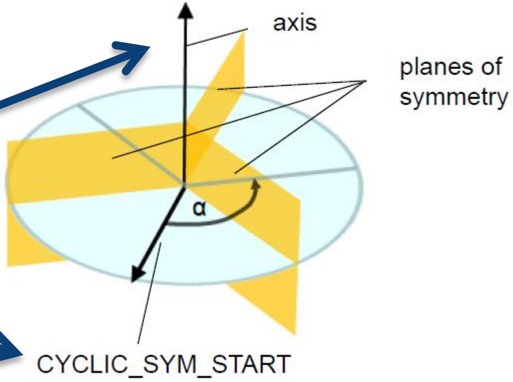
```
! ---------------------------------------
!
! Task: Task-1
!
OPTIMIZE
  ID_NAME = Task-1
  DV = Task-1_DESIGN_AREA_
  OBJ_FUNC = MIN_MAX_STRESS
  DVCON = GR_FIXED

  !---COPY BELOW THIS POSTITION ---!



  STRATEGY = SHAPE_CONTROLLER
  MESH_SMOOTH = Task-1_MESH_SMOOTH_
END_
```

Example | Wind turbine hub

Step 8: Definition of a symmetry restriction within the parameter file

```
!----------------------
!
!  Link Condition: LINK_SURF_CYCLIC_PLANE_SYM
!
LINK_SHAPE
  ID_NAME = LINK_SURF_CYCLIC_PLANE_SYM
  MASTER = MAX
  CLIENT = SURF_CYCLIC_PLANE_SYM
  CYCLIC_SYM_START =  0., 0. , 1.
  CLIENT_DIR = 1.,0.,0.
  CS = CS_0
  TOL = 0.1
  ANGLE = 120.
END_
```

$\alpha = 120°$

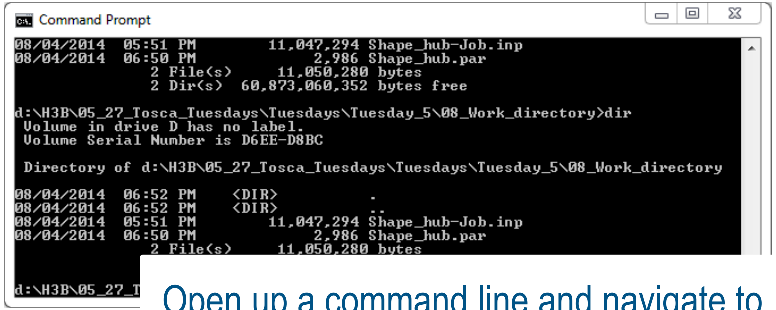


Further information in the Tosca Structure documentation, page 2-345

Example | Wind turbine hub

Step 9: Submission of the optimization task

In order to submit the optimization job the batch mode of Abaqus/CAE is used within this example



Open up a command line and navigate to the defined work directory where the Abaqus input deck (*.*inp*) and the Tosca optimization file (*.*par*) are located.

Example | Wind turbine hub

Step 9: Submission of the optimization task

Use the following command line to submit the Tosca optimization from batch mode.

```
abq6134 optimization –j Shape_hub –task Shape_hub
```

OR

```
abaqus optimization –j Shape_hub  –task Shape_hub
```

The Tosca optimization should then start and an optimization directory called "*Shape_hub*" is created.

Example | Wind turbine hub

Step 9: Submission of the optimization task

The status of the optimization can be checked through the outputfile *“TOSCA.OUT”.* It is located in the *“Shape_hub”* directory.

```
(INFO    ) [18:57:08|atom_perl]
(INFO    ) [18:57:08|atom_perl]     T o s c a    f o r    A b a q u s
(INFO    ) [18:57:08|atom_perl]
(INFO    ) [18:57:08|atom_perl]     Version Abaqus 6.13-4
(INFO    ) [18:57:08|atom_perl]     Build: 80_131219-010015
(INFO    ) [18:57:08|atom_perl]    Copyright(c) 2013 by FE-DESIGN GmbH and Dassault Systemes, all rights reserved
(INFO    ) [18:57:08|atom_perl]
(INFO    ) [18:57:08|atom_perl] Logging to STDOUT at loglevel NOTICE
(INFO    ) [18:57:08|atom_perl] Logging appended to log-file "Shape_hub\TOSCA.OUT" at loglevel INFO
(NOTICE  ) [18:57:08|atom_perl] Logfile: Shape_hub\TOSCA.OUT
(INFO    ) [18:57:08|atom_perl] Tosca for Abaqus Time information: 04.08.2014 at 18:57:08
```

...

```
(INFO    ) [19:06:46|atom_opt] Application finished successfully
(INFO    ) [19:06:46|atom_perl]  Elapsed time of atom_opt.CMD          : 00:00:12  -  current time: 19:06:46
(INFO    ) [19:06:46|atom_perl]  Total Elapsed time of Tosca for Abaqus : 00:09:38  -  current time: 19:06:46
(INFO    ) [19:06:46|atom_perl] Tosca for Abaqus Time information: 04.08.2014 at 19:06:46
(INFO    ) [19:06:46|atom_perl] Logfile: Shape_hub\TOSCA.OUT
(NOTICE  ) [19:06:46|atom_perl] Tosca for Abaqus ended successfully
```

The optimization finishes after ~10 minutes (4 CPUS).

Example | Wind turbine hub

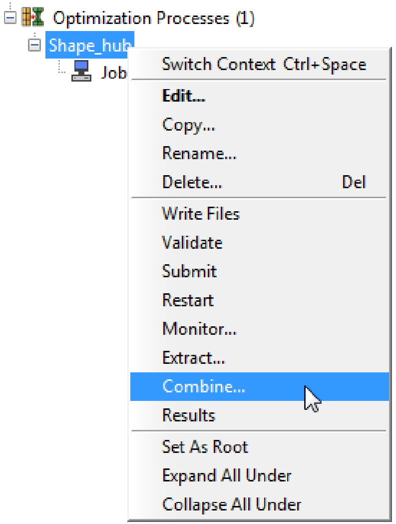
Step 9: Submission of the optimization task

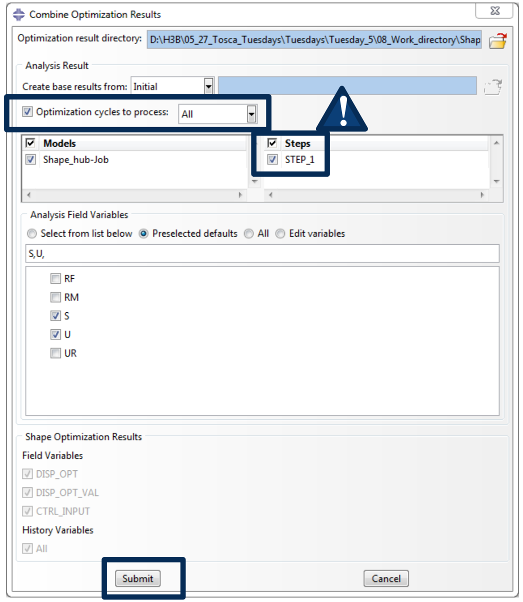
The status of the objective function and other design responses can be checked in the *"optimization_report.csv" file.*

	A	B	C	D	E
1	ITERATION	MIN_MAX_STRESS	OBJ_FUNC_DRESI	OBJ_FUNC_TERM:DRES	
2	Norm-Values:	56672.81	82315.05	56672.81	
3	0	56672.81	82315.05	56672.81	
4	1	52058.05	76313.32	52058.05	
5	2	52019.06	75931.41	52019.06	
6	3	50922.51	74525.66	50922.51	
7	4	49070.2	72408.71	49070.2	
8	5	46899.95	69993.7	46899.95	
9	6	46303.93	69169.37	46303.93	

Example | Wind turbine hub

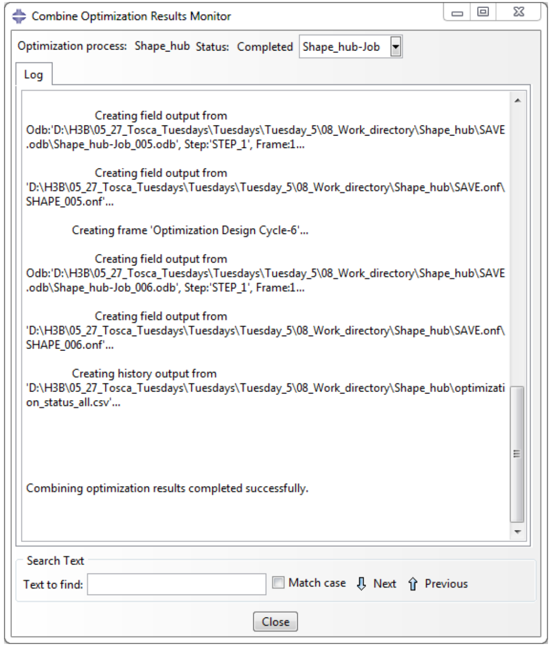
Step 10: Visualization in Abaqus/CAE (⚠ Optimization has to be completed)

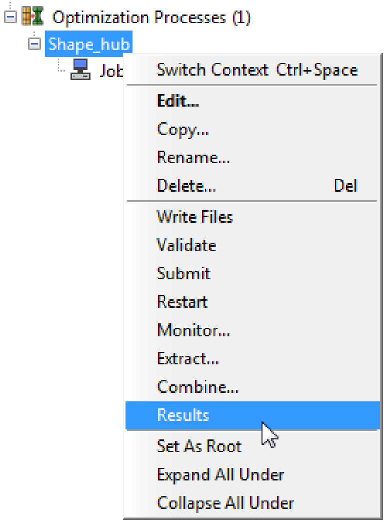




Example | Wind turbine hub

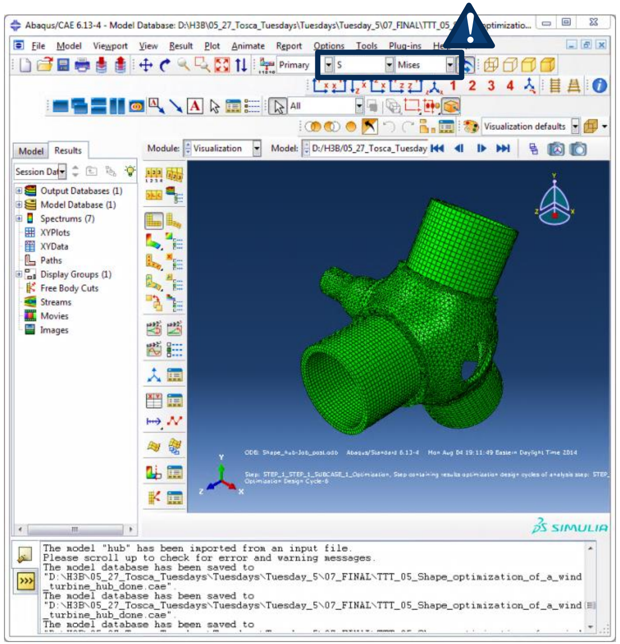
Step 10: Visualization

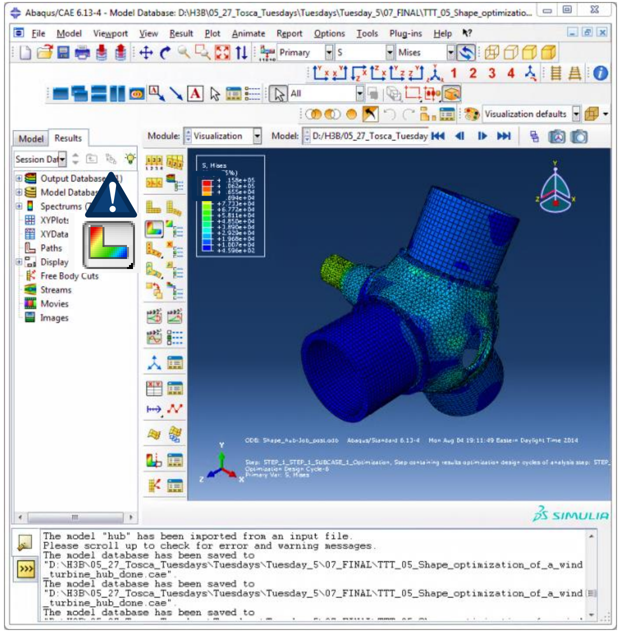




Example | Wind turbine hub

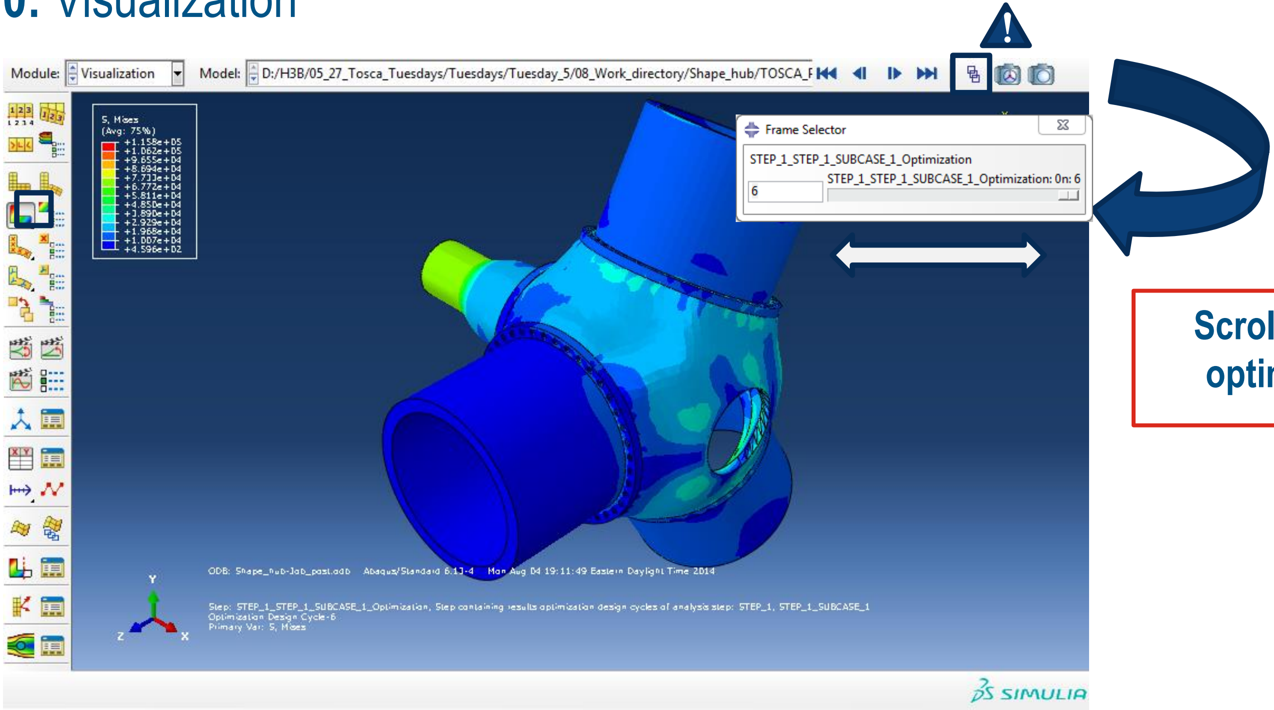
Step 10: Visualization





Example | Wind turbine hub

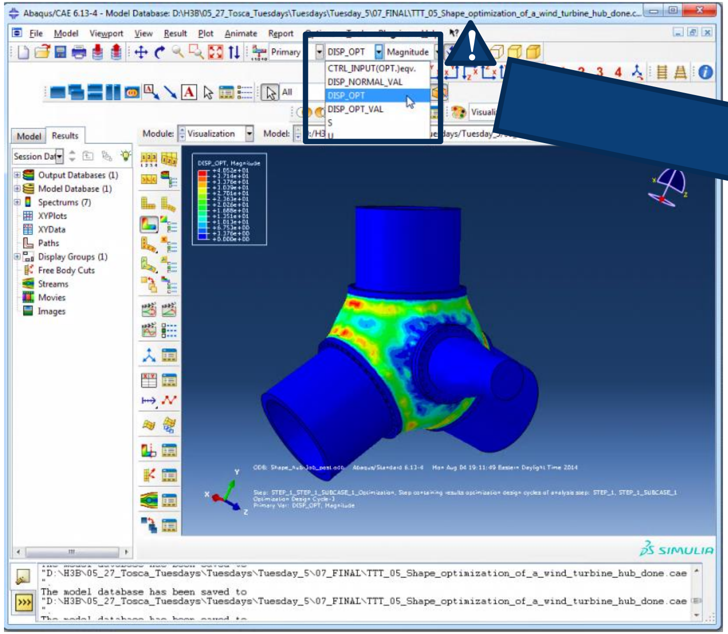
Step 10: Visualization

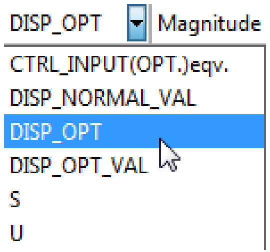


Scroll through each optimization cycle

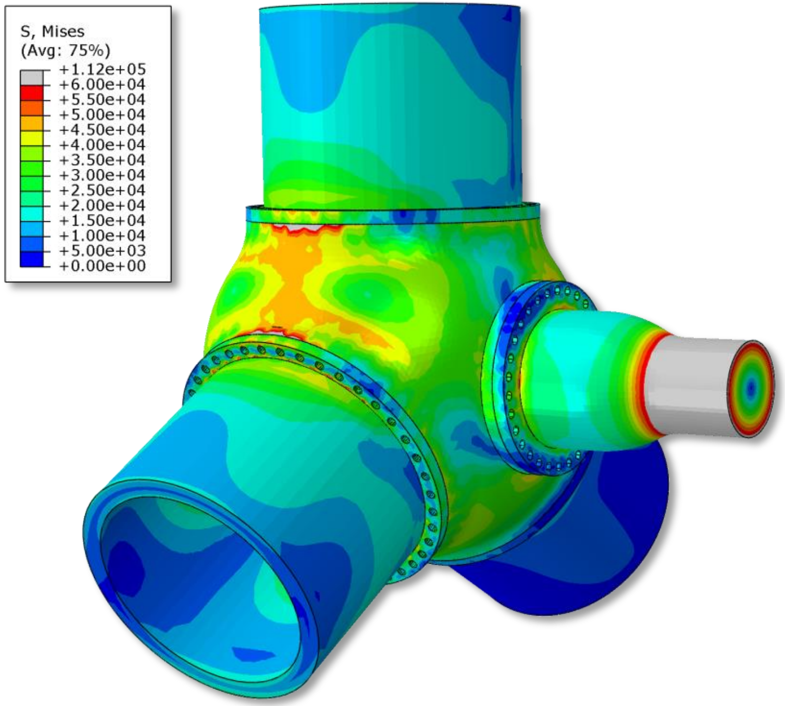
Example | Wind turbine hub

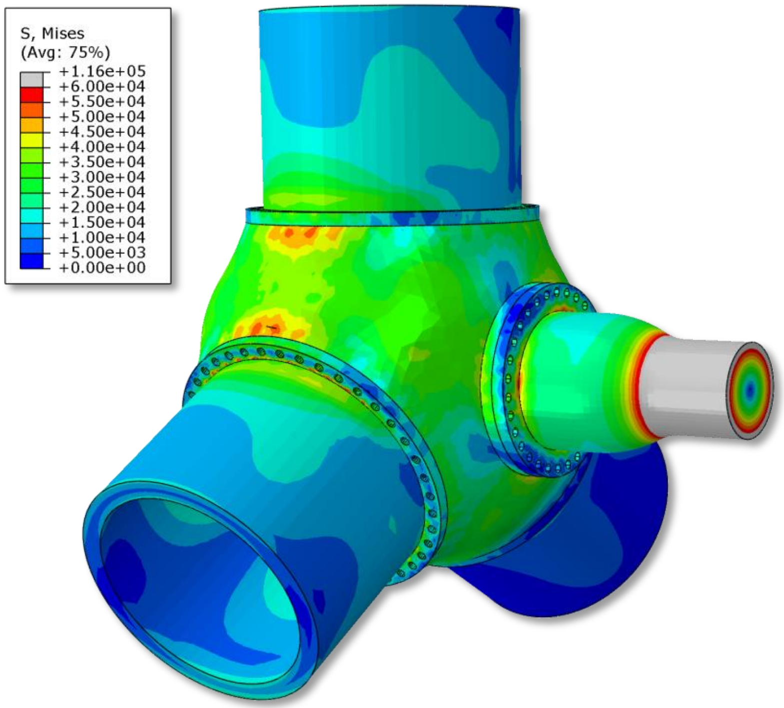
Step 10: Visualization





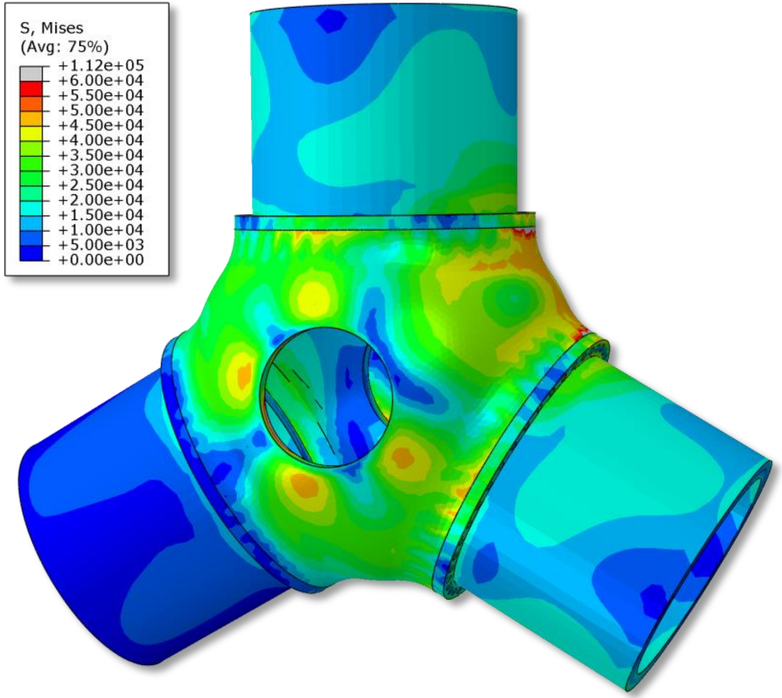
Optimization displacements of design nodes



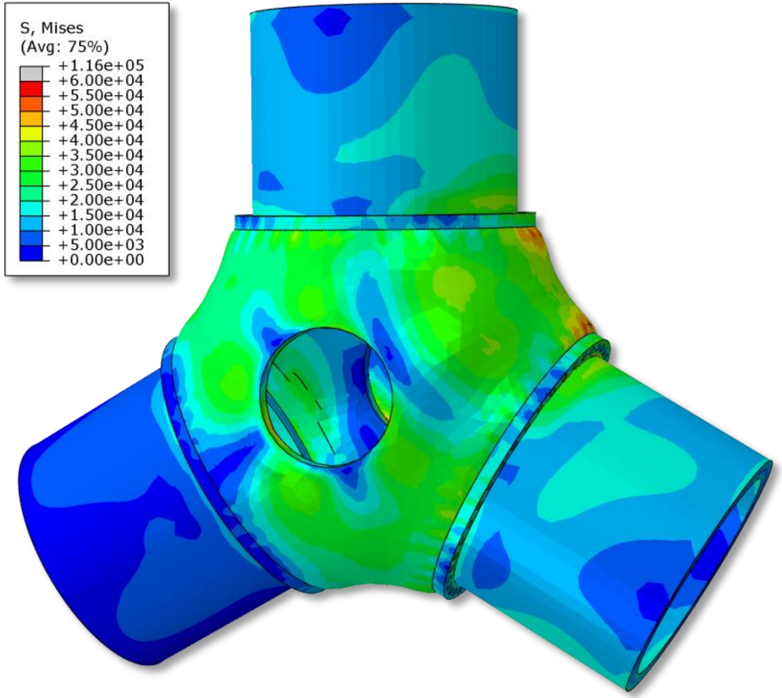


Original Design

Optimized Design



Original Design



Optimized Design

IF WE ™