TOPOLOGY OPTIMIZATION process description using Abaqus ATOM

Topic: Introduction to topology optimization using Abaqus ATOM

Name: Steffen Johnsen

Date: 10.04.2013

Approved by: Terje Rølvåg

Abaqus ATOM is one of the most powerful commercial software today for topology optimization.

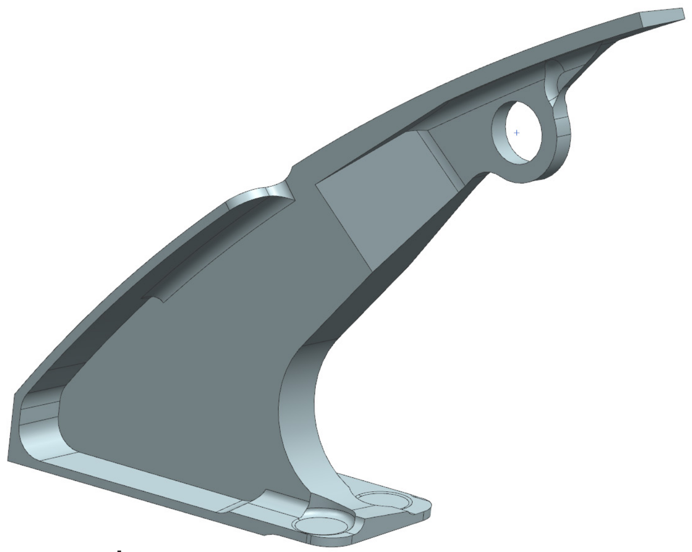
This knowledge brief is aimed at providing knowledge transfer and understanding of the process involved with setting up and understanding the results of a topology analysis.

As a Single Objective Analysis software with capabilities of specifying weighted objectives, Abaqus ATOM can solve a vast amount of different problems, including, but not limited to problems of structural, thermal, flow, electrical and magnetic nature.

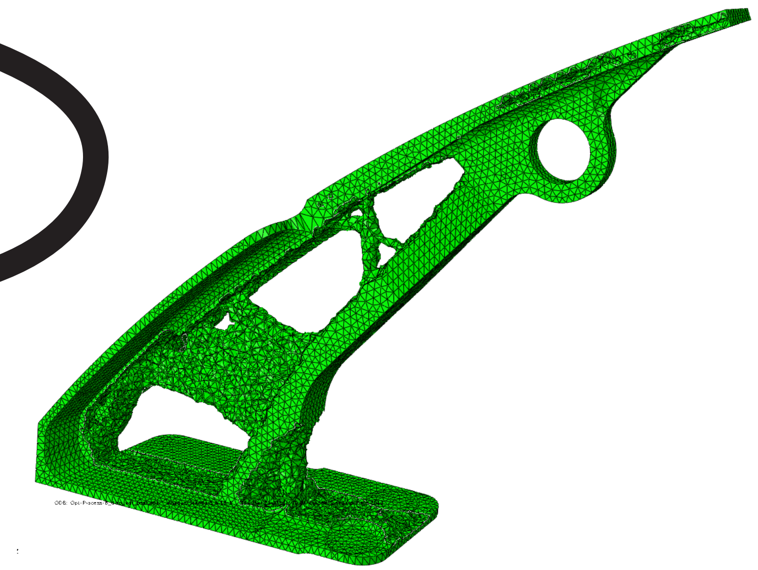
The process description assumes that the reader is familiar with the UI of the CAE- system and is capable of setting up a standard linear structural simulation. The reader should also possess basic knowledge of FE- theory.

Topology optimization is particularly powerful for enhancing creativity and visualizing solutions not initially apparent to the engineer. The method is unique as is provides optimal solutions for the given boundary conditions and specified design responses. Engineering experience will always be required to evaluate results, but is naturally limited when faced with conflicting constraints and the ability to find optimal solutions.

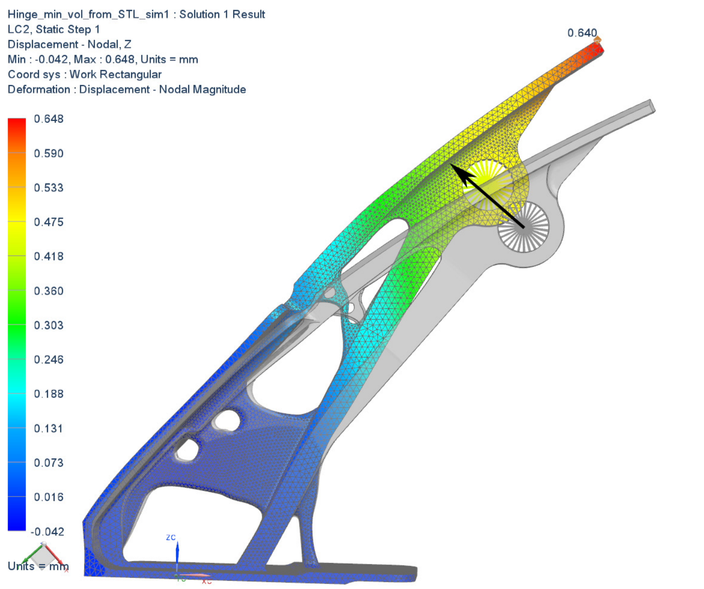
Original part



Design space



Optimization result



Reference simulation

Reengineered part

suplight

SIMULIA

UGS The PLM Company

modeFRONTIER

NTNU

TOPOLOGY OPTIMIZATION process description using Abaqus ATOM

Topic: Creating a new topology optimization

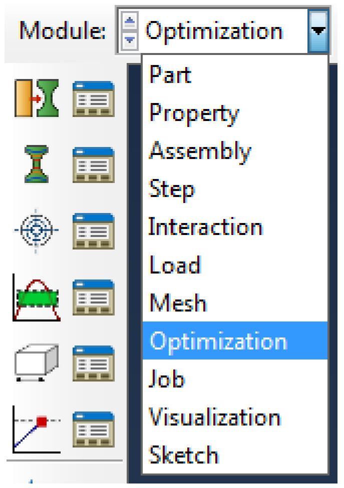
Name: Steffen Johnsen

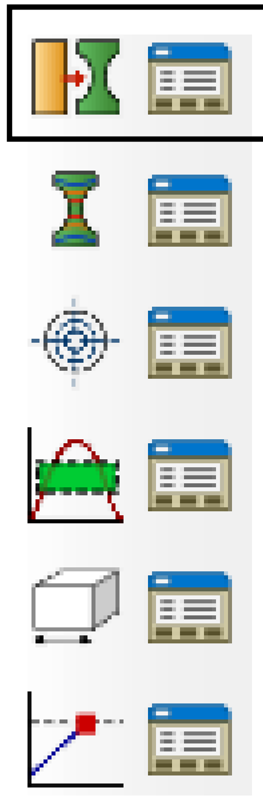
Date: 10.04.2013

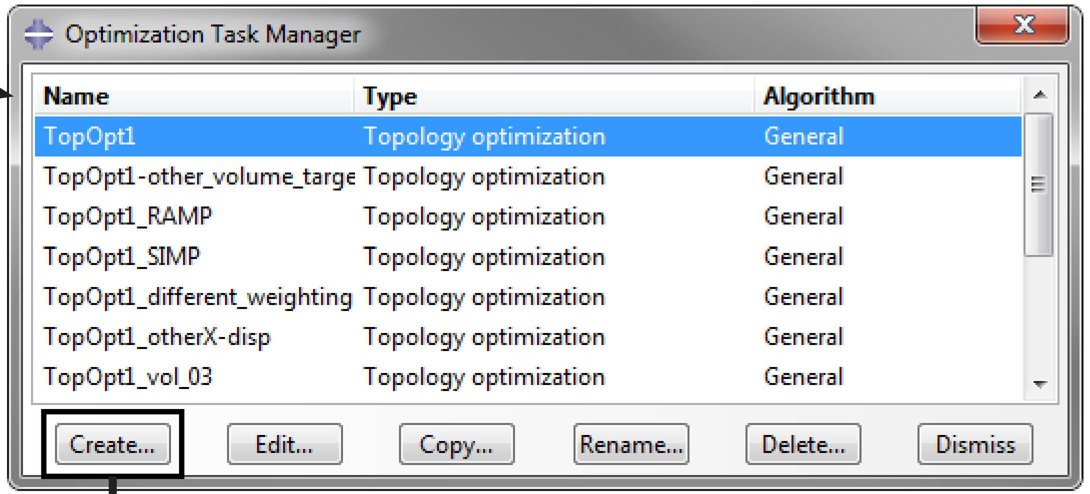
Approved by: Terje Rølvåg

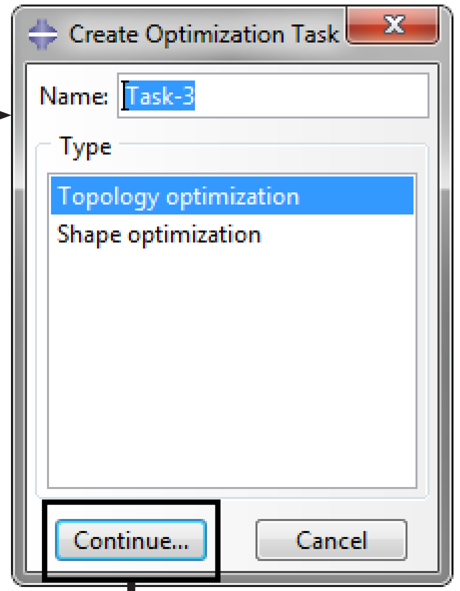
To create a new topology optimization simulation, setup a the model under analysis with properties, boundary conditions and mesh as you would do with any other FE- simulation.

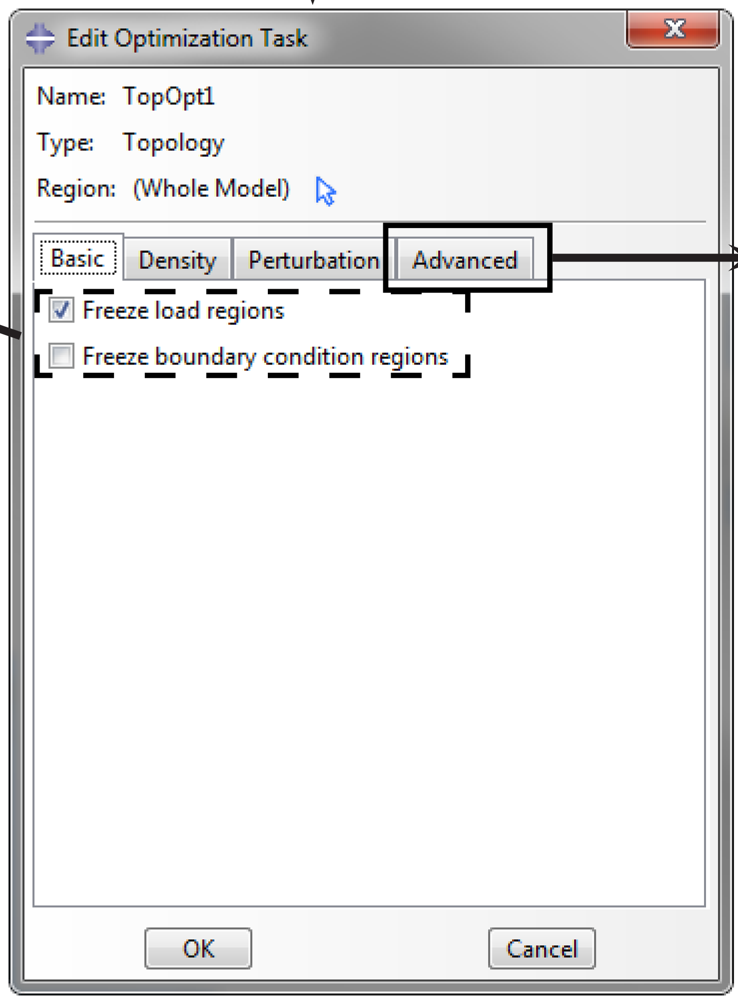
Then navigate to Module -> Optimization and navigate through the menus you can see on the next pages.



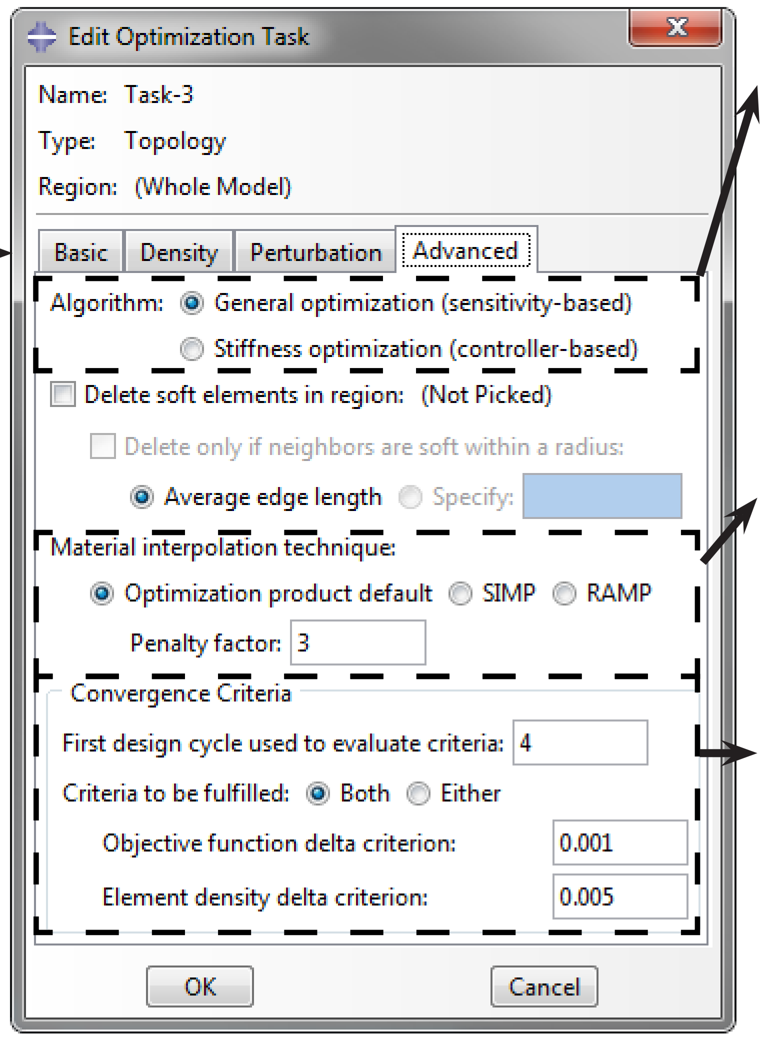








If you are performing a pressure load analysis, freezing load regions will eliminate the need for using the RAMP material interpolation technique. Additonal frozen regions can be specified at a later stage.



For problems where only strain energy and volume are of interest, the more efficient stiffness optimization algorithm can be used.

Use the default material interpolation technique (normally the SIMP technique will be used when set to default) unless dealing with problems containing pressure loads, where the RAMP technique is the better choice when boundary faces are changing during the optimization.

Convergence criteria used to dermine if the solution has converged. Default values are quite strict, so for initial simulations, corser values might be used to lower simulation time. Additional convergence criteria can also be specified at a later stage.

suplight

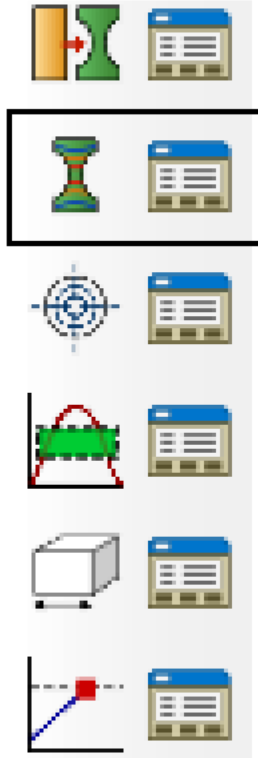
TOPOLOGY OPTIMIZATION process description using Abaqus ATOM

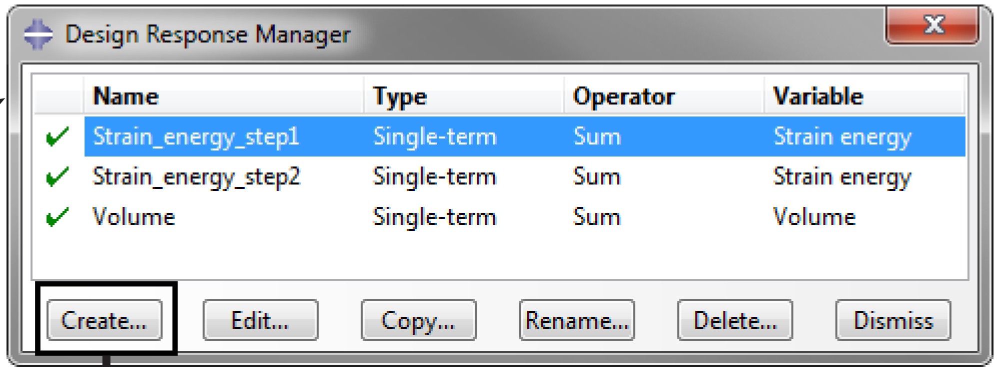
Topic: Defining design responses for the optimization	Name: Steffen Johnsen	Date: 10.04.2013	Approved by: Terje Rølvåg

Design responses can be seen as the simulation variables. They are defined before the simulation objective and constraints are set.

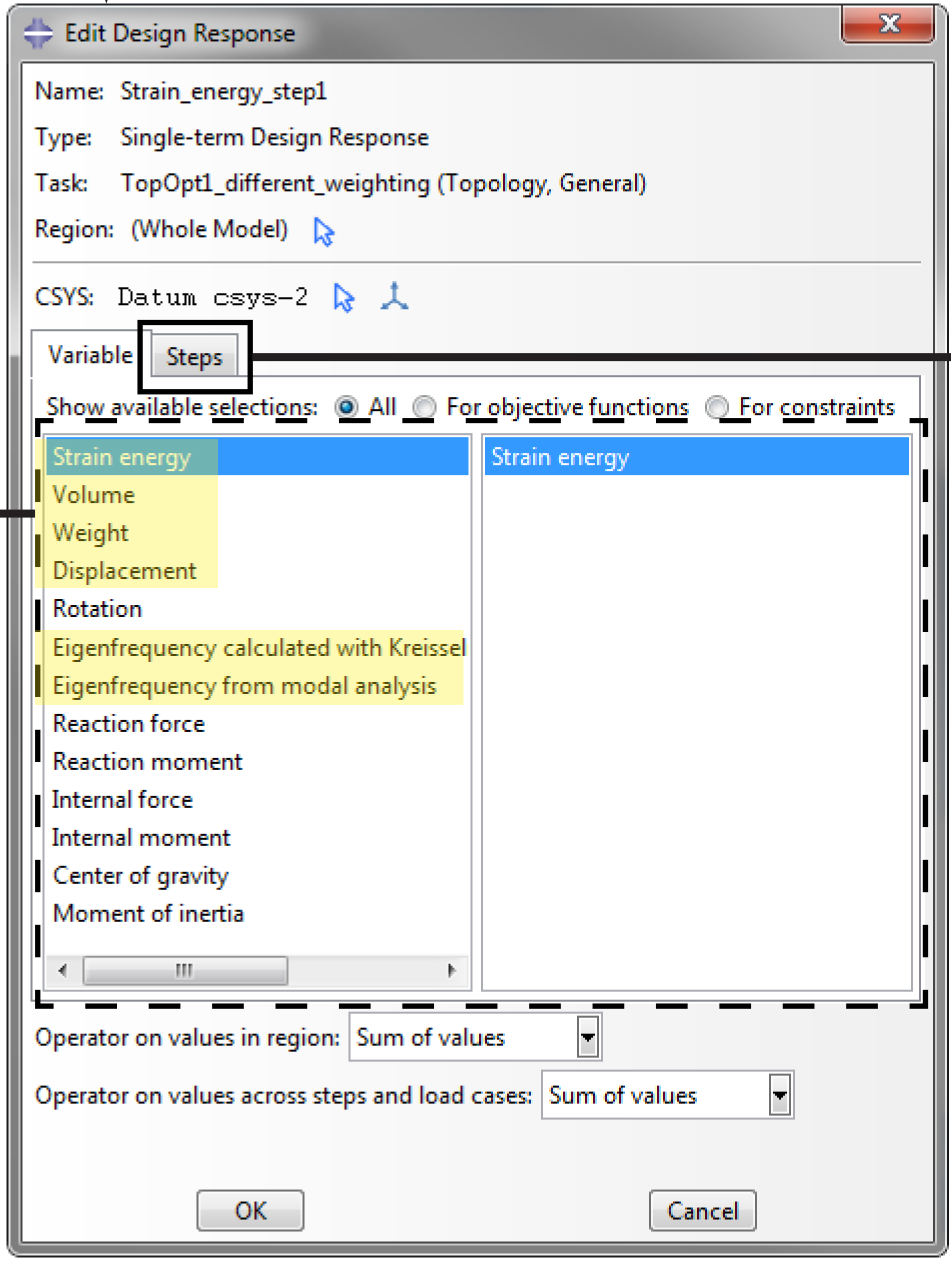
Design responses may be specified for the entire model, or for specific regions of the geometry.

Best practice when defining design responses:
- If possible, define the design responses for smaller areas. These areas might be specific regions of interest, such as fillets, holes and functional surface. The more specific the area or region, the better the simulation will run, leaving less chance of detecting other mechanisms than the one intentded.
- Use the Tools -> Datum -> Point or CSYS to define control points or additional coordinate systems for easier reference of loads and boundaries and specification of design responses. When defining a displacement constraint, it is important to specify the coordinate system and direction of interest.

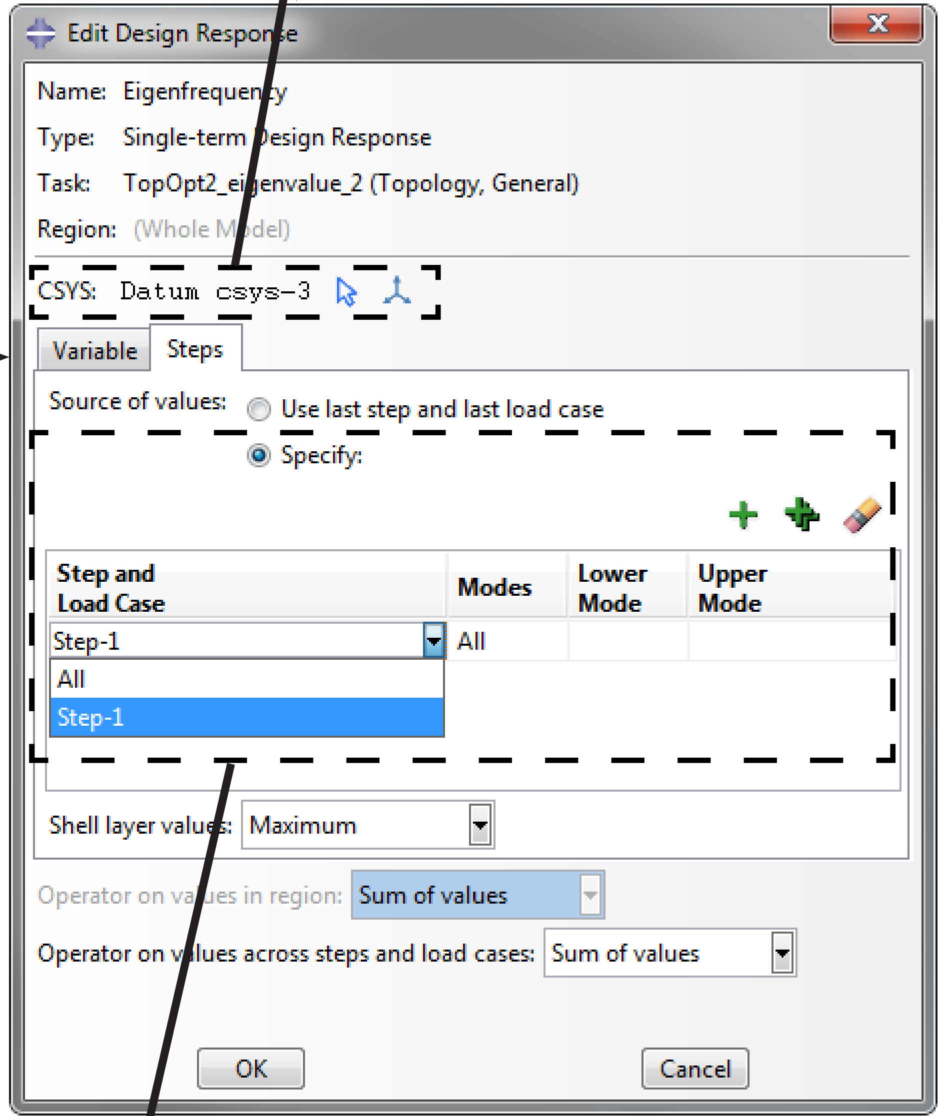




Specifying the coordinate system is vital when optimizing wrt. displacement in a particular direction (ex. the load direction), as these directions are selected as principal axis in the coordinate system of choice.



Design responses that are available for the chosen optimisation algoritm.



If you have multiple load cases in different simulation steps, specifying the step(s) for which the design response is valid is important in order to get correct results.

suplight

UGS The PLM Company · modeFRONTIER · NTNU

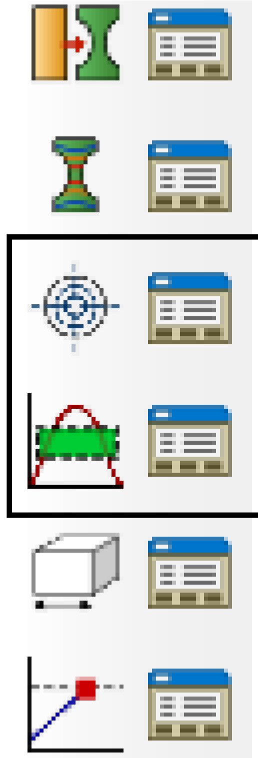
TOPOLOGY OPTIMIZATION process description using Abaqus ATOM

Topic: Defining the objective definition and constraints | Name: Steffen Johnsen | Date: 10.04.2013 | Approved by: Terje Rølvåg

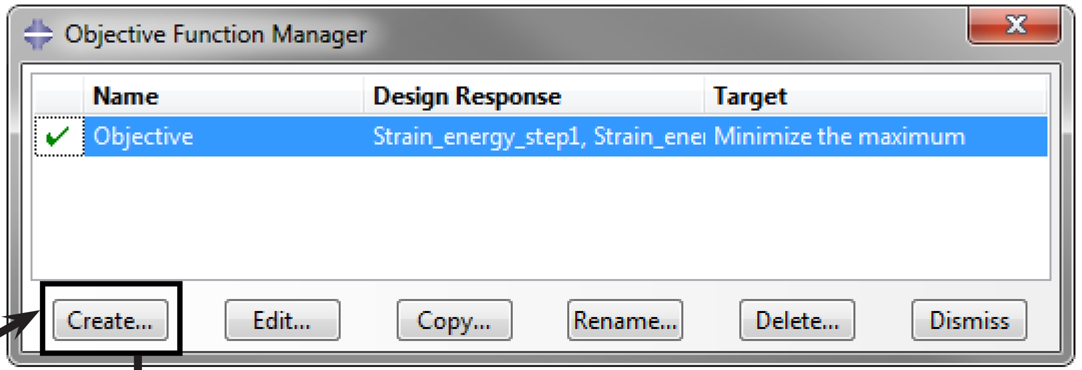
In general, there are three different approaches to optimizing a structure using Abaqus:

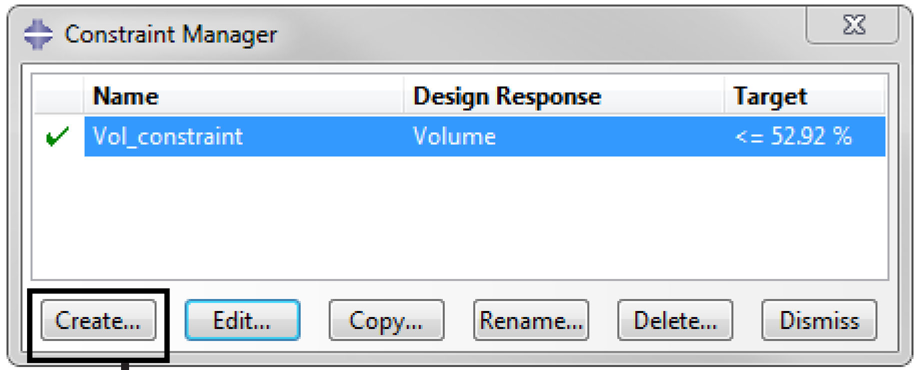
1) Minimize (or minimize the maximum) strain energy, constrained with an upper limit on the relative volume fraction.
2) Minimize the volume, constrained with an upper limit for displacement, moment of inertia, reaction forces, CoG or rotation (or a combination of these).
3) Maximize eigenfrequencies, constrained with a LOWER limit for volume or/and limits for other design responses that are independent of loads (as eigenvalue analysis are performed without external forces applied).

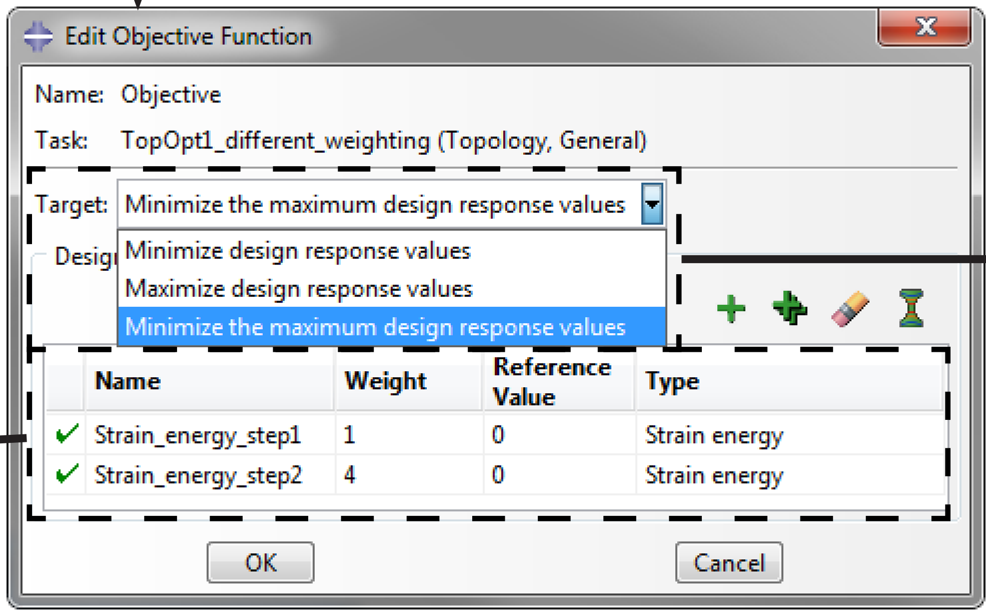
Best practice when defining the objectives and constraints:
- Choose the design response that has the highest uncertainty. If the target volume is specified, choose an approach that has volume as a constraint.
- If you have more than one load case, the MinMax formulation will provide a solution that has better overall robustness.



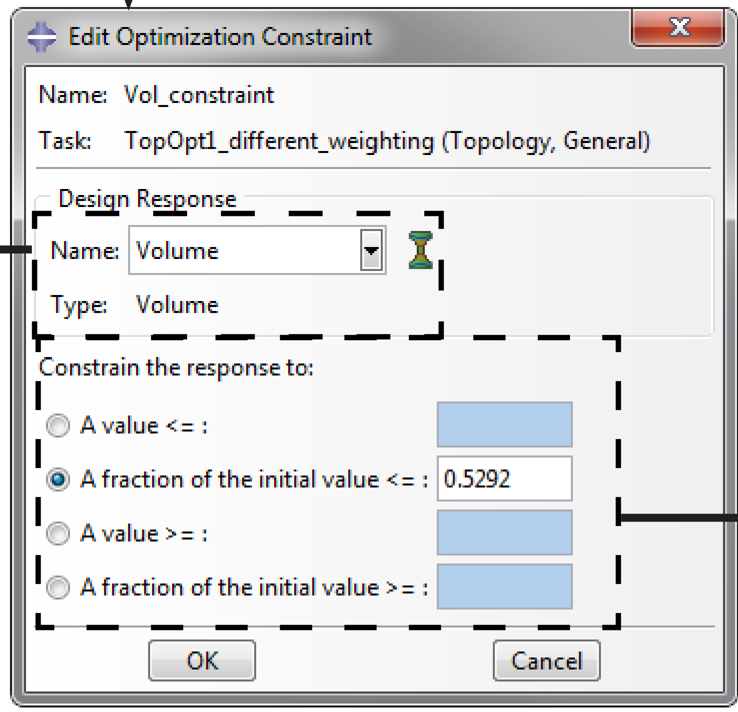
If the target type is similar for more than one design response, a weighted average can be specified.





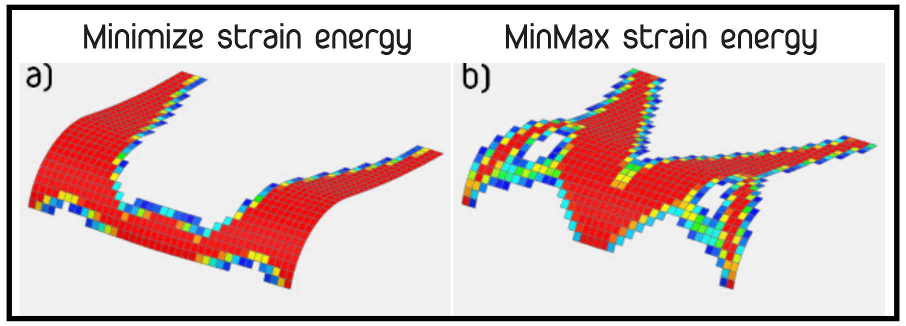


The objective target can be set to MINIMIZE, MAXIMIZE or MINIMIZE THE MAXIMUM (MinMax). F. ex. you will most likely minimize a volume objective, maximize an eigenvalue objective or minimize the maximum strain energy (requires that you have multiple design responses with strain energy defined, one for each load case). The MinMax formulation is especially powerful for creating robust designs, as the final solution will be equally good for alle load cases.



Specification of which design response(s) to use as simulation constraints.

Constraints can be specified as an absolute value or as a relative value, both for upper or lower limits. Multiple constraints can be specified. Be aware that more constraints will decrease the chance of converging the objective.



suplight

UGS The PLM Company | modeFRONTIER | NTNU

TOPOLOGY OPTIMIZATION process description using Abaqus ATOM

Topic: Defining geometric constraints | Name: Steffen Johnsen | Date: 10.04.2013 | Approved by: Terje Rølvåg

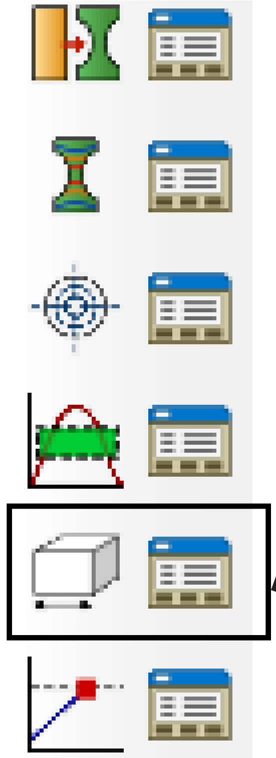
As a rule of thumb, a general analysis should be performed before geometric constraints are introduced, as they will increase computational time as well as limit the available design space, giving a less optimal solution.

Demold control is probably one of the most important geometric constraints, as it ensures that production methods like casting, stamping, forging and machining can be used. If the design space BC's introduce some kind of torsion or angular momentum in the part, internal voids and holes will be made, as material is preferred at the outer premises of the design space. This is done to increase the second moment of inertia.

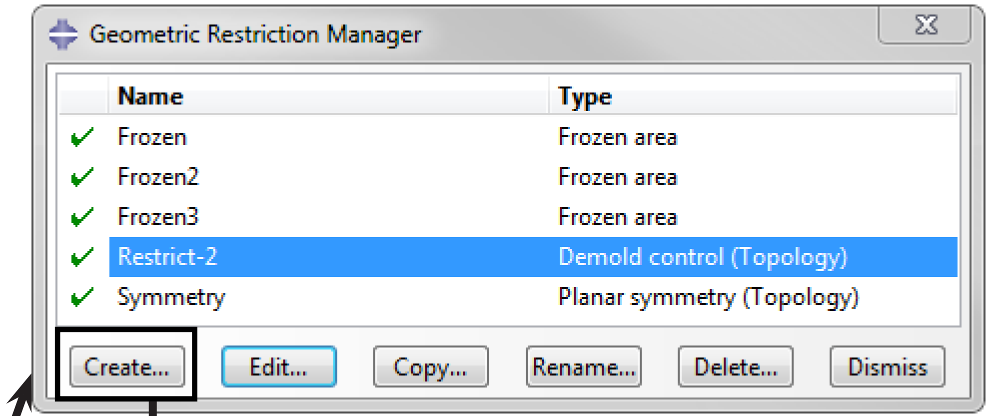
Symmetry constraints can be specified as planar, rotational, cyclic or for a point. Using the symmetry constraint, solutions will look more aesthetic, and they will be easier to produce.

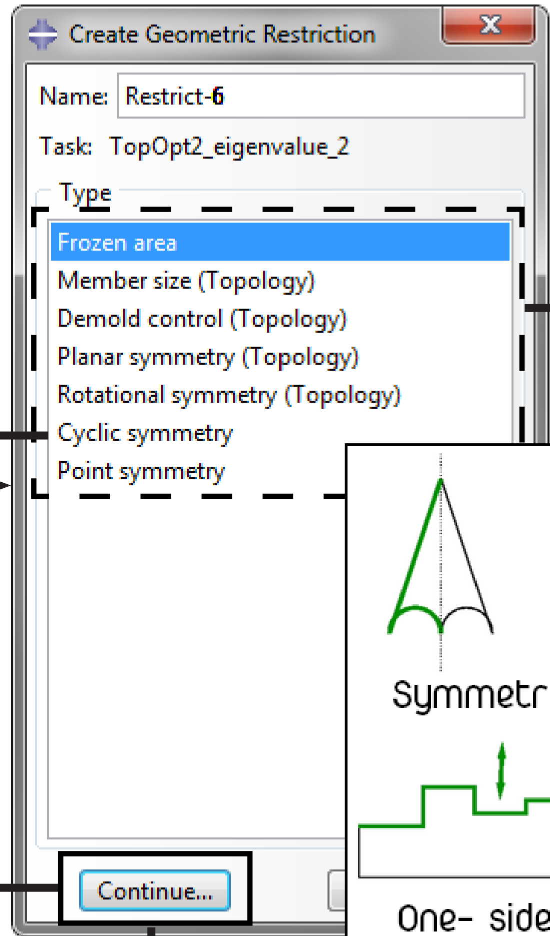
Frozen faces will not be affected, and will remain the same after the analysis has ended. This is particularly useful to maintain areas of geometric significance, as the algorithm is forced to shave off weight from other areas of the design space.

Specifying an lower or upper limit for the member size will restrict the results from having too thin og thick members that cannot be produced.

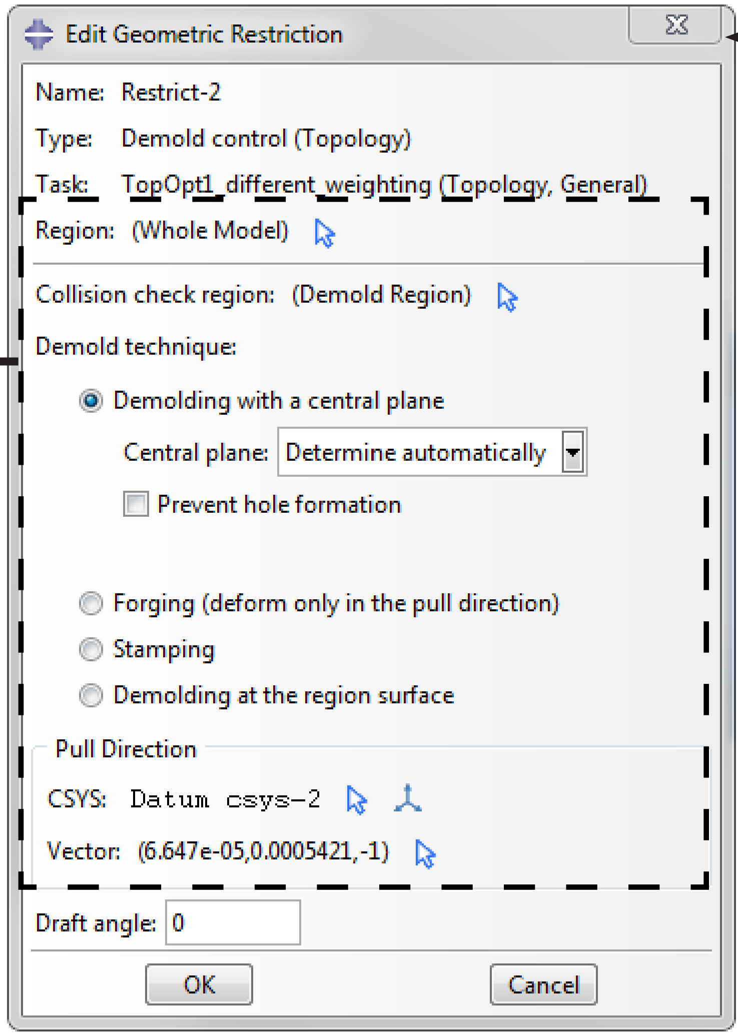


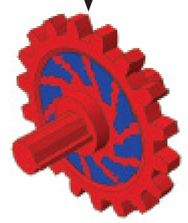
The designer must specify which region to control, which region to perform collision checks on (between tool and workpiece), as well as type of demold technique , coordinate system and pull direction vector.

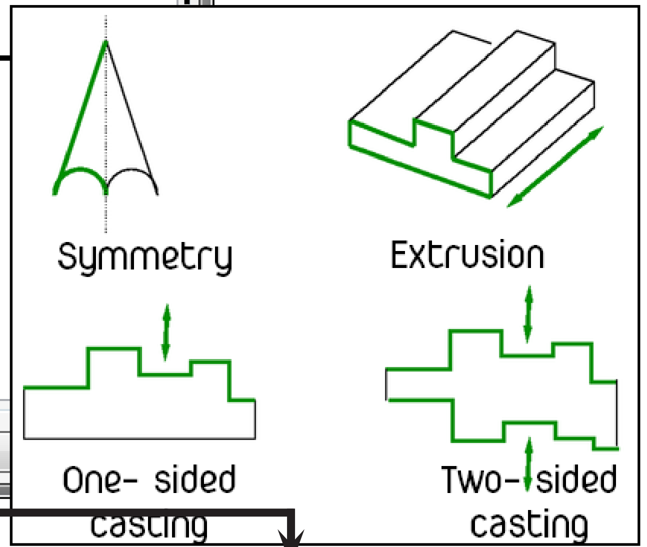


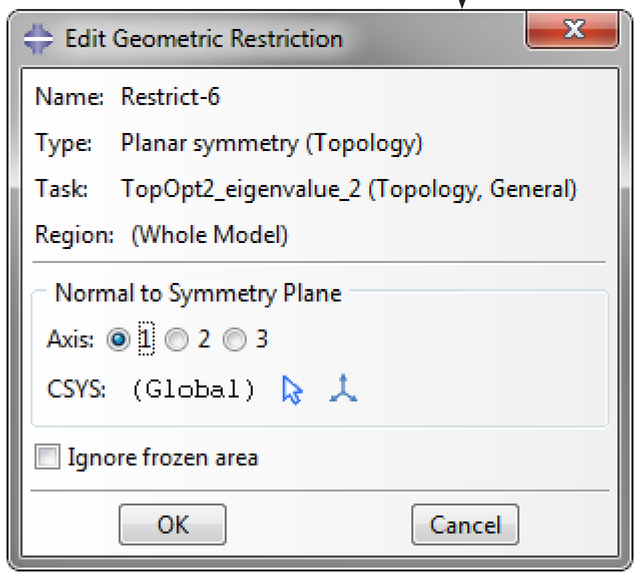


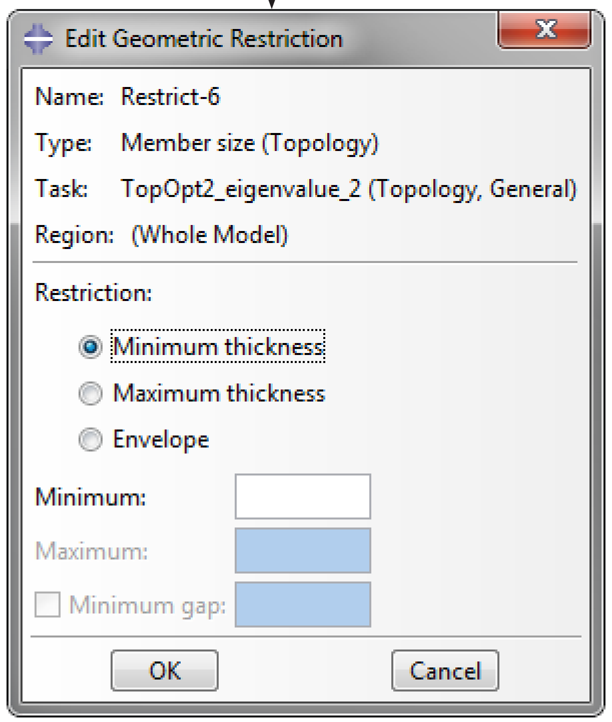
Available geometric restrictions.











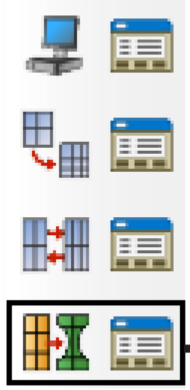
suplight

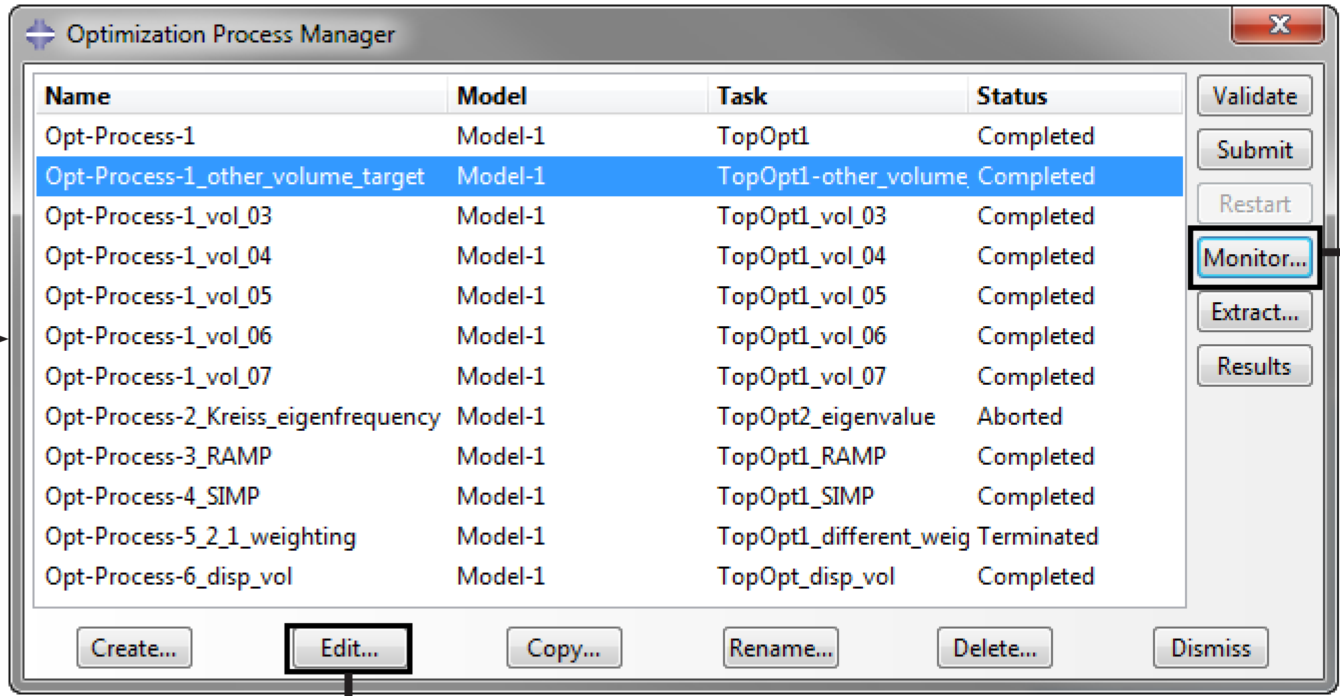
UGS The PLM Company | modeFRONTIER | NTNU

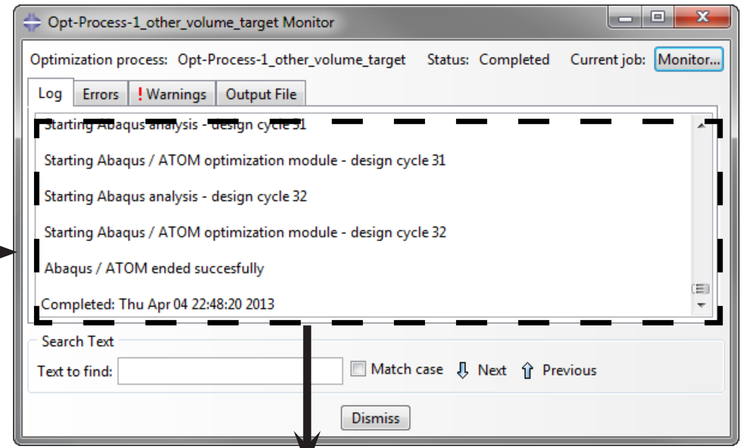
TOPOLOGY OPTIMIZATION process description using Abaqus ATOM

Topic: Running the topology optimization | Name: Steffen Johnsen | Date: 10.04.2013 | Approved by: Terje Rølvåg

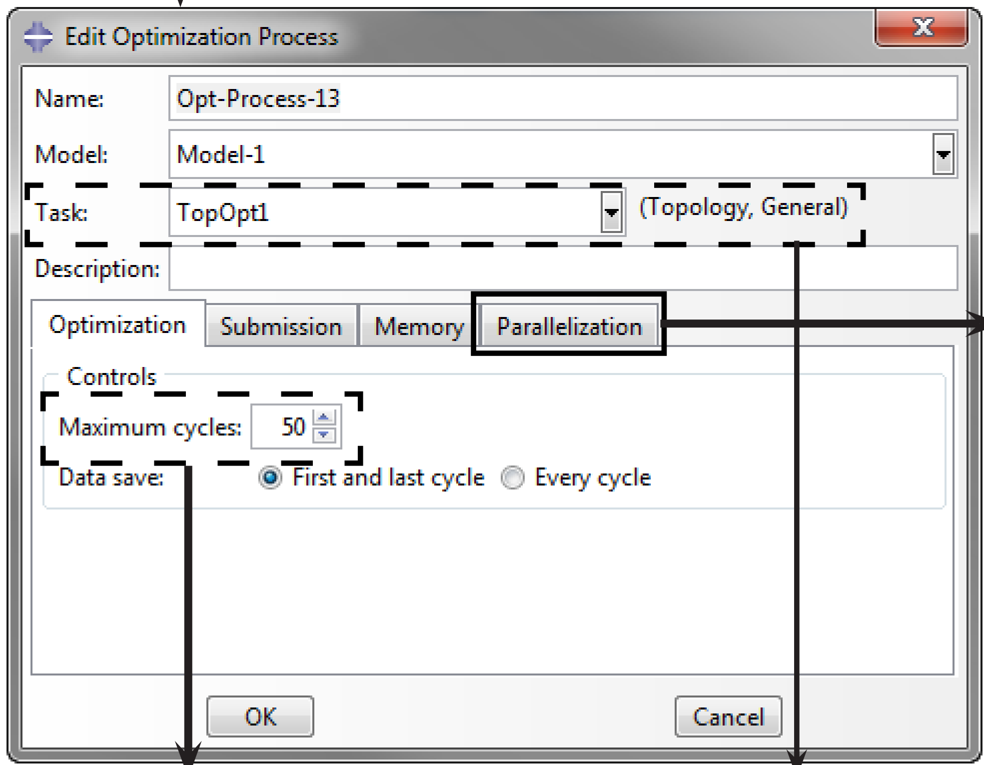
In order to run the simulation, an optimization process must be made. In the optimization Process Manager; Clicking Monitor will let you see the progress for each iteration, while clicking Results will let you see the iterations so far, visualized on the part.

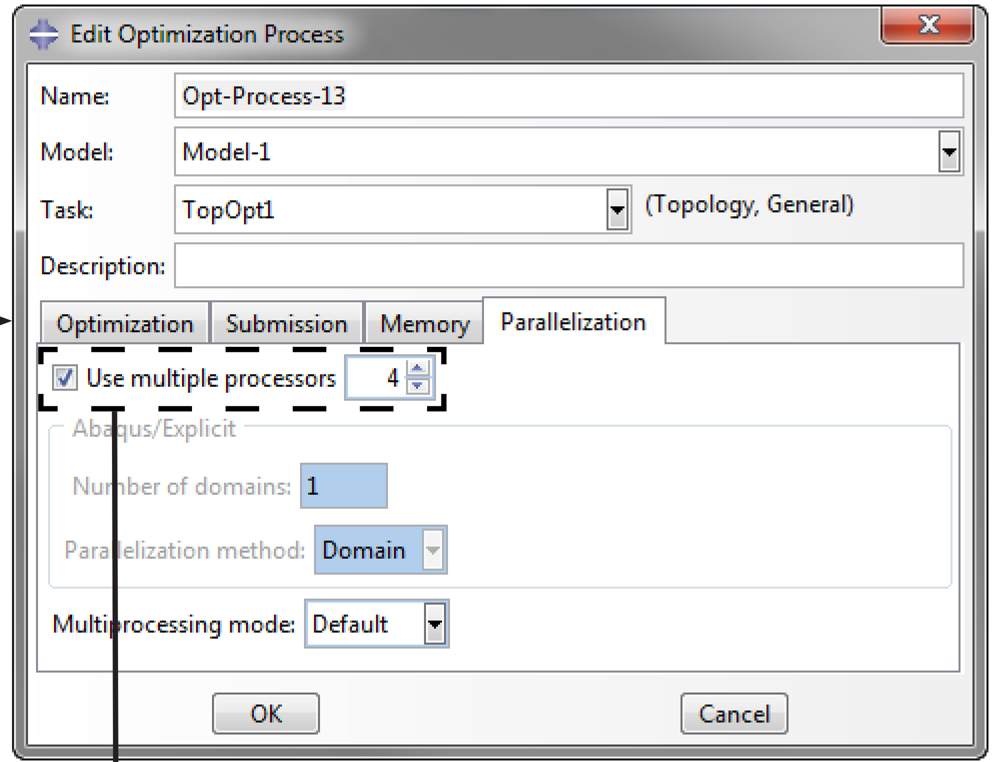






When monitoring the simulation, you can see that the algorithm performs a density update followed by a FE- simulation to update the design response values and check for convergence.





The maximum number of cycles can be limited, but the number should be so high that the convergence criteria stops the simulation.

Choose which topology optimization to solve.

If you have more than two processors, assign more to speed up the simulation.

suplight

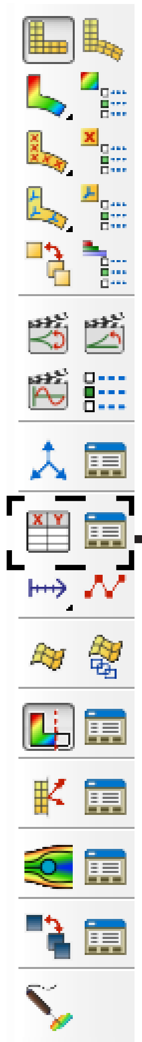
UGS The PLM Company | modeFRONTIER | NTNU

TOPOLOGY OPTIMIZATION process description using Abaqus ATOM

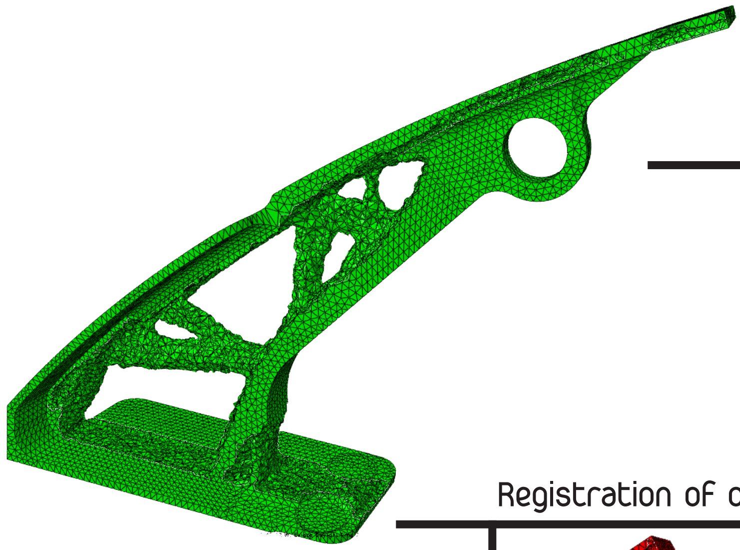
Topic: Post- processing of optimization results | Name: Steffen Johnsen | Date: 10.04.2013 | Approved by: Terje Rølvåg

Post- processing of results is important to verify that the simulation has converged, and that the design constraints are within limits. Graphs can be generated by the software by clicking the Graph-icon in the visualization manager (see illustration below), by selecting the design responses of interest and plotting these.

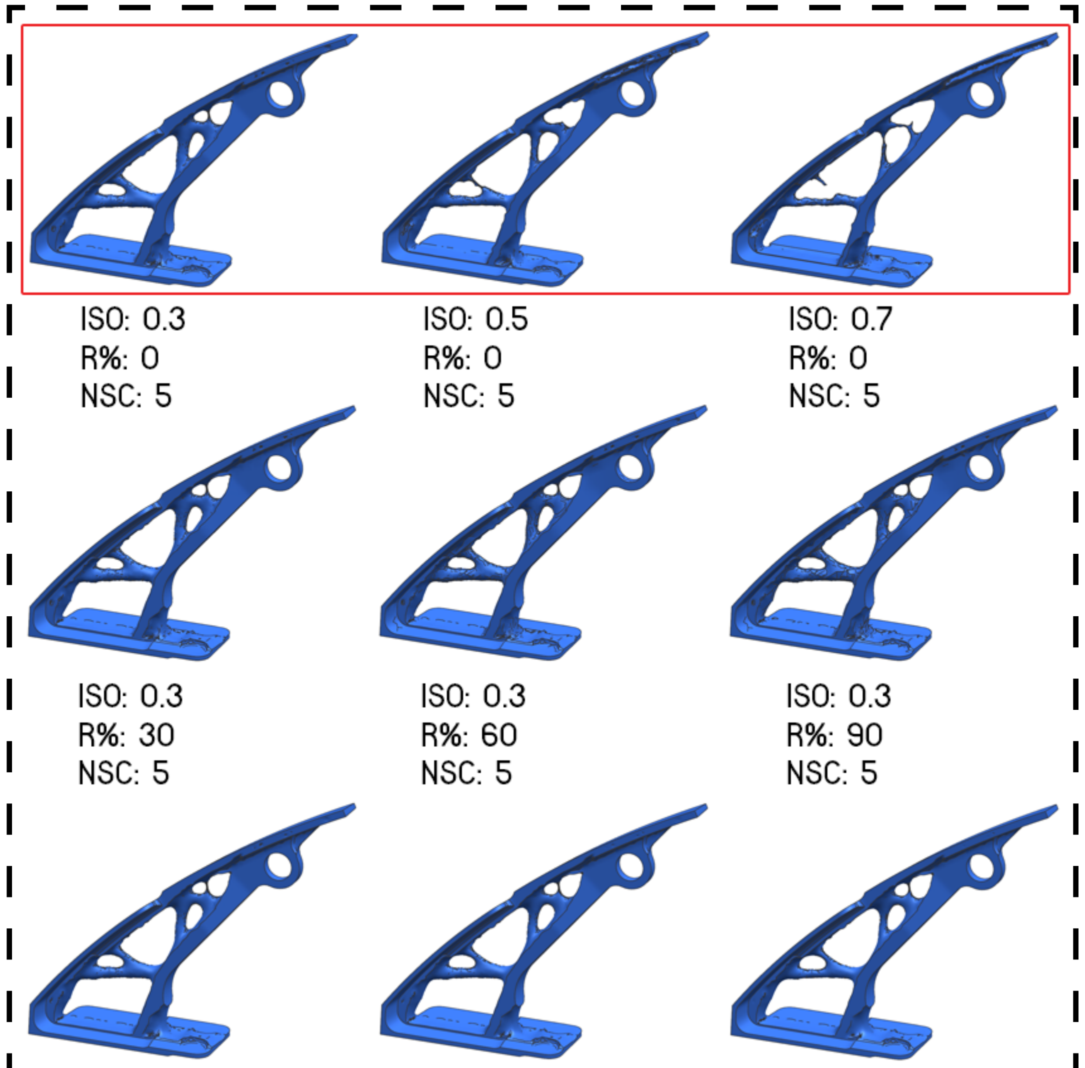
When viewing results that are dependent on direction, the user can specify which coordinate system to be used by clicking (in Visualization environment) Results -> Options -> Transformation -> User- specified, and choosing the correct CSYS of interest. Results surface extraction can be done by clicking (in the Job environment) Optimization Process Manager -> Extract. As the figure in the upper right corner shows, varying the ISO- value (or Target volume) will have the largest impact on the resulting STL- file. Increasing the Number of Smoothing Cycles (NSC) will give a smoother surface, while increasing the Reduction percentage (R%) will give a corser surface as the surface complexity will be reduced.



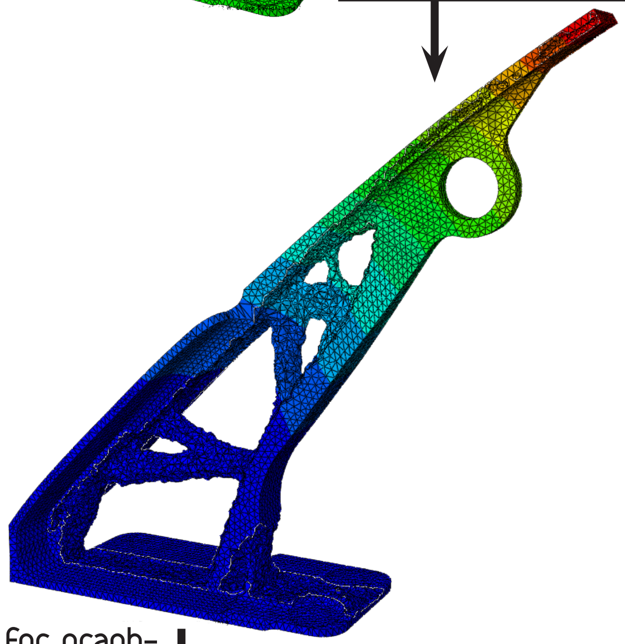
Tools for graphing the design responses.

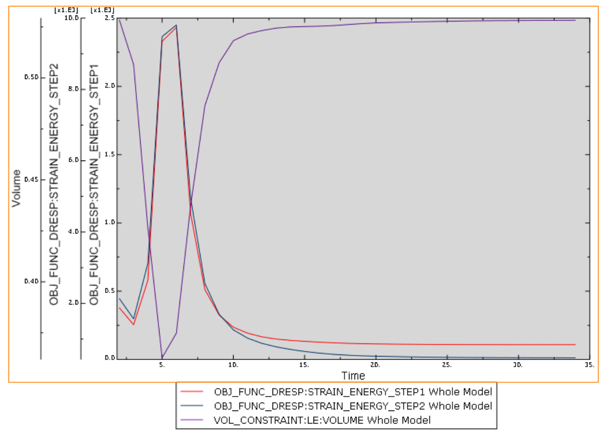


Surface extration (STL files)

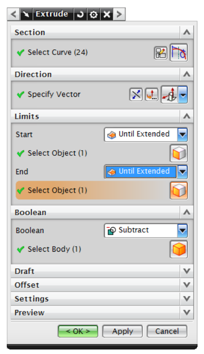


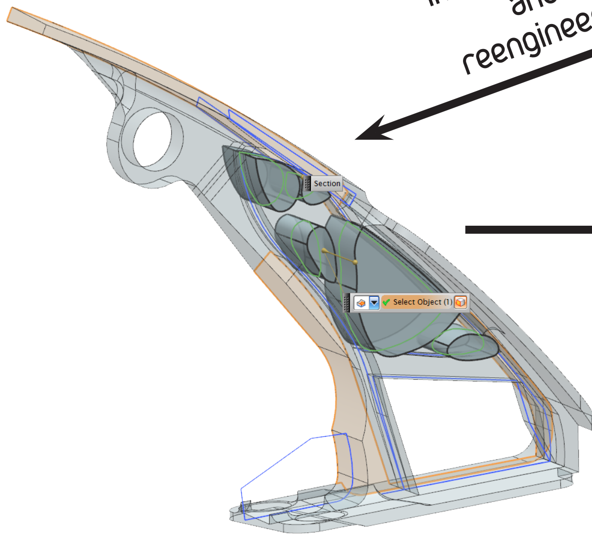
Registration of optimization results

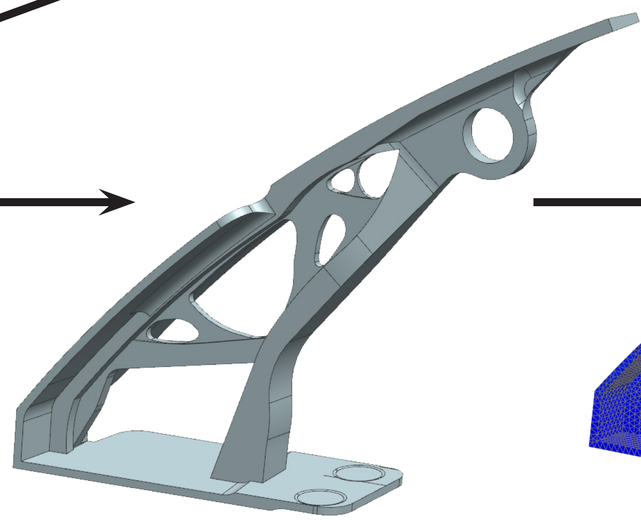


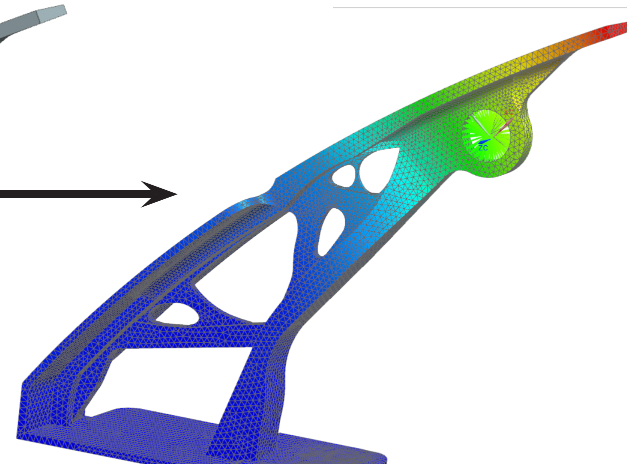


Interpretation and reengineering









suplight

UGS The PLM Company | modeFRONTIER | NTNU

TOPOLOGY OPTIMIZATION process description using Abaqus ATOM

Topic: Topology optimization; minimize strain energy with upper volume limit, summary

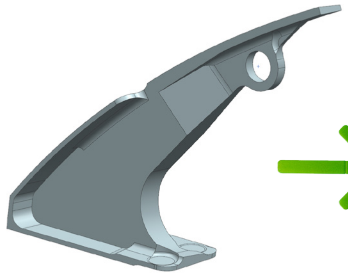
Name: Steffen Johnsen

Date: 15.04.2013

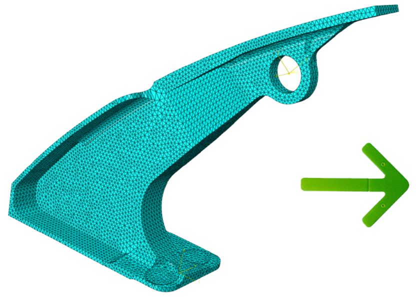
Approved by: Terje Rølvåg

1. Setup a normal static analysis containing all boundary conditions. Both linear analysis and geometric nonlinear analysis are supported.
2. Evoke the Optimization environment, and create a new Optimization task, and define a name for the optimization. Make sure to choose whether to use the General algorithm or the Stiffness algorithm.
3. Define all Design responses in the Design Response Manager (all properties that will be influenced in the analysis; volume, eigenfrequencies, strain energy, displacement, etc.)
4. Choose which Design response(s) to use as an Objective in the Objective Function Manager. Weigh multiple objectives against each other and choose whether to maximize, minimize or minimize the max (preferred).
5. Define scalar constraints for the remaining Design responses.
6. Define Geometric Constraints (symmetry, frozen areas, cast condition, etc.). Use as few as possible, and run the first optimization without to see what is demanded.
7. Create an Optimization process in the Job Manager and specify which topology optimization to run. Start the optimization process.
8. Monitor progress by clicking Monitor during the optimization process in the Job Manager
9. When optimization has converged: Look at results by clicking Results in the Job Manager.
10. In the Job Manager: Click Extract and extract an STL- file from the final result. Make a copy of the .prt- file with the design space, import the STL file and use modeling tools and transform the part to look like the optimization result, or import the STL into an empty part and build the new part from scratch.

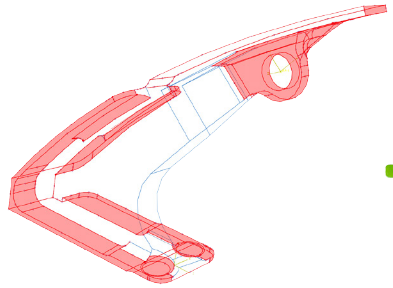
Original geometry



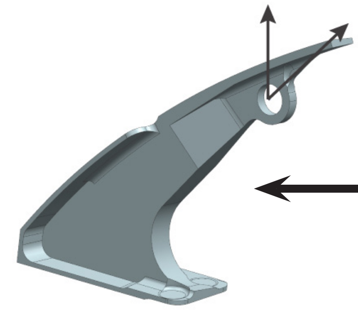
Design space



Meshed model

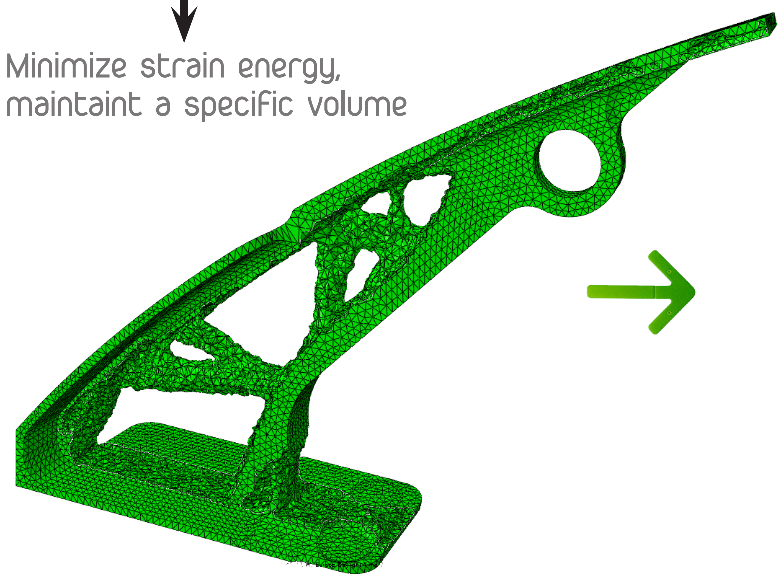


Model w/ geometric constraints

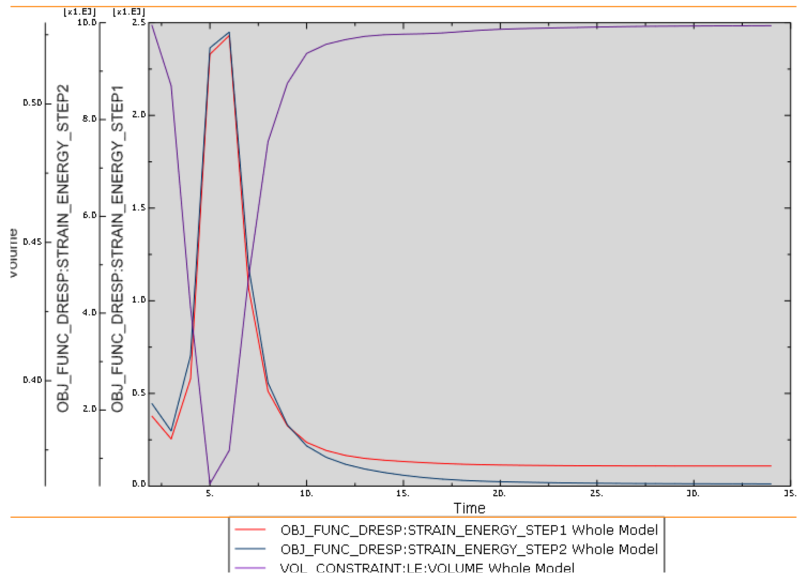


Loaded model (2.5 kN LC1 and LC2 loads defined in subsequent steps). Constrained at the bottom surface.

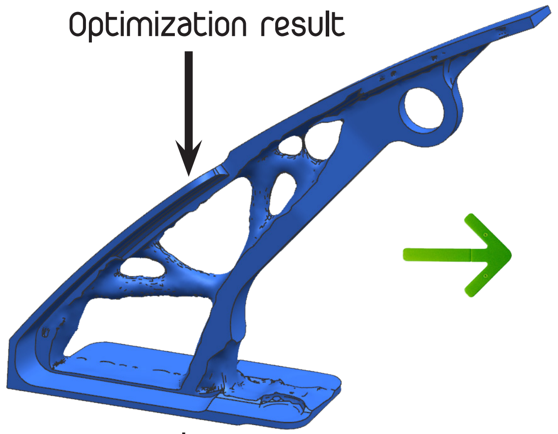
Minimize strain energy, maintaint a specific volume



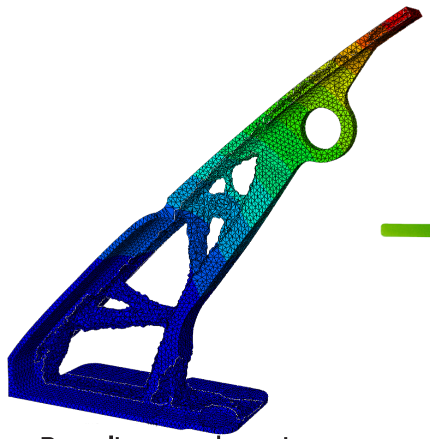
Optimization result



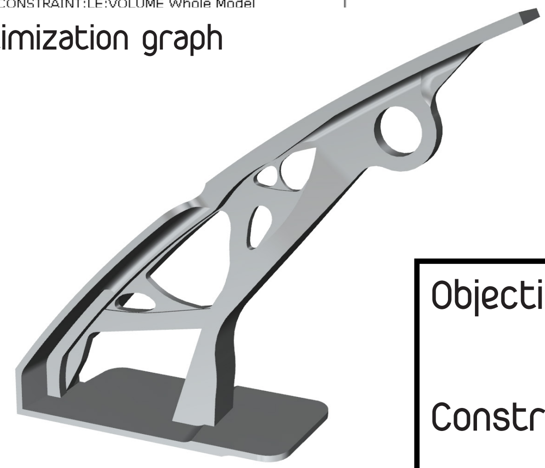
Optimization graph



Extracted geometry



Results read- out



Reengineered part

Objective:	Minimize strain energy (max. stiffness) Weighting of load cases 1 : 1
Constraints:	Save 10% volume -> Volume fraction
Geometric constraints:	Frozen areas (see figure), symmetry (not for frozen areas) and forging (no inner voids).

suplight

WEIGHT SAVED: 10% (OPTIMIZATION RESULTS)

CHANGE IN STIFFNESS (OPT. RESULTS):

TENSILE: - 26.36%

BENDING: + 5.59%

UGS The PLM Company modeFRONTIER NTNU

TOPOLOGY OPTIMIZATION process description using Abaqus ATOM

Topic: Topology optimization; MDO with displacement and strain energy vs. volume

Name: Steffen Johnsen

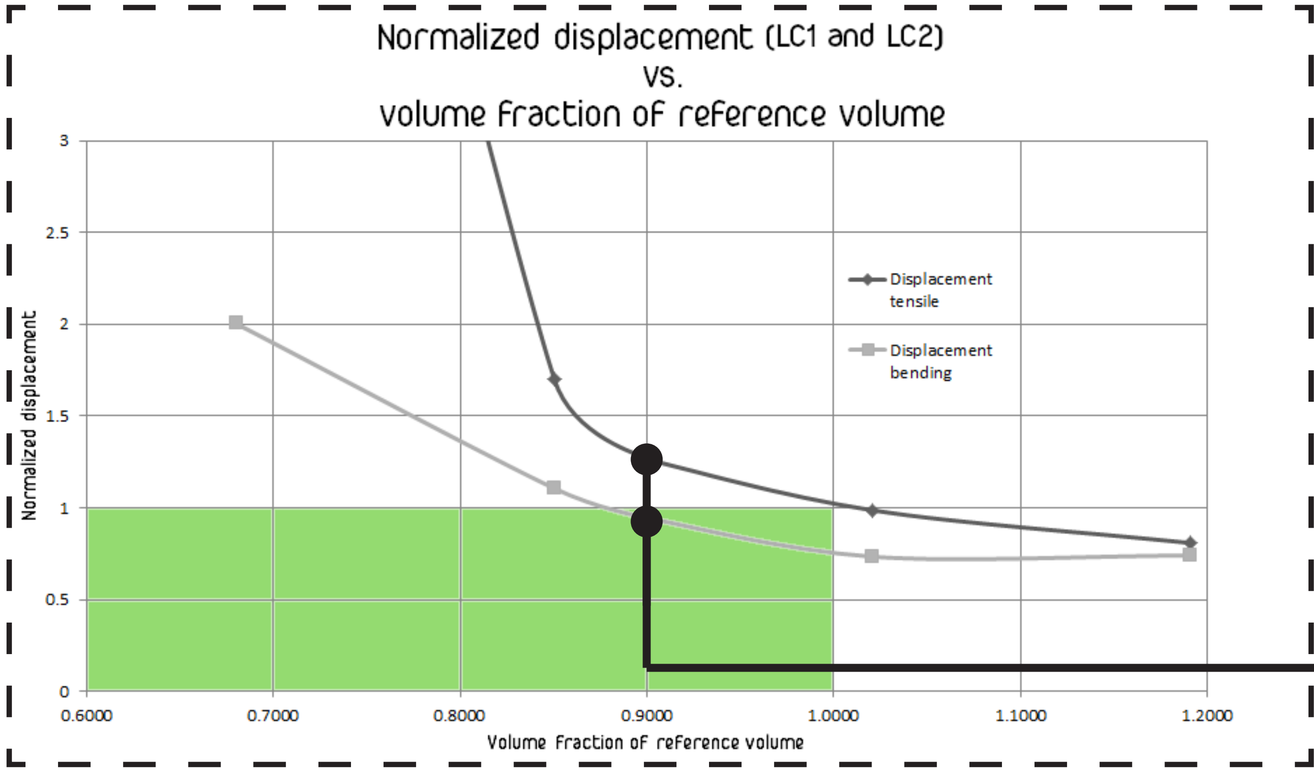
Date: 05.04.2013

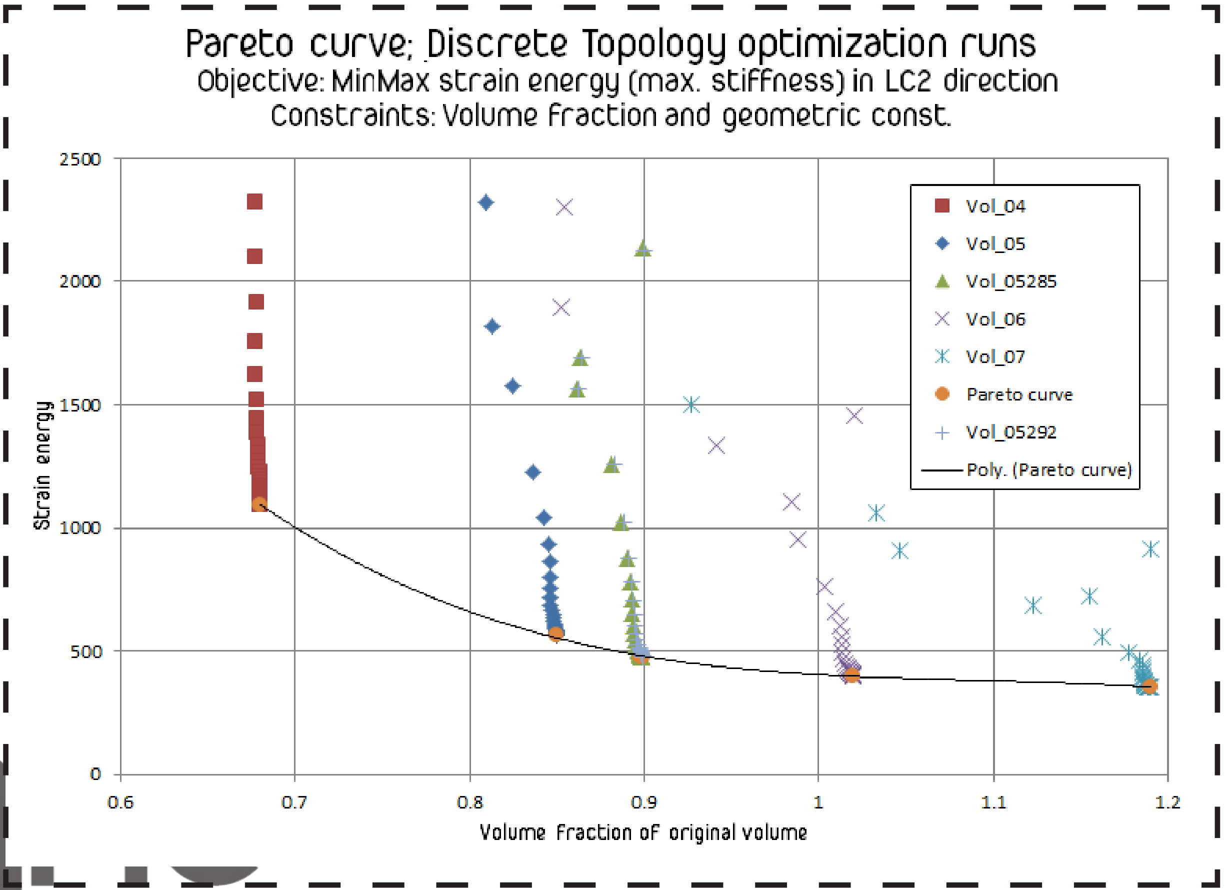
Approved by: Terje Rølvåg

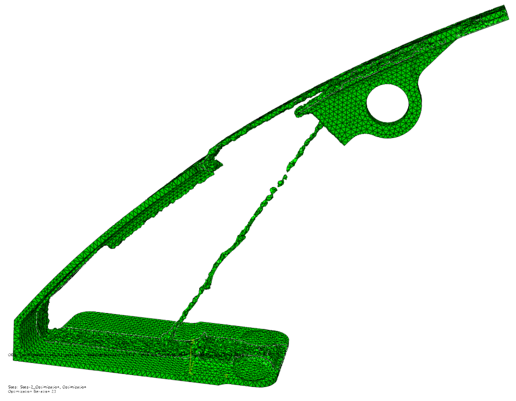
Using a MDO (multi- disciplinary optimization)- approach to the simulation makes it possible to display Pareto curves.

The simulations have been performed with the objective to MINIMIZE STRAIN ENERGY, hence maximize stiffness, as well as maintaining a volume less than a SPECIFIED VOLUME CONSTRAINT. As both NX and Abaqus are able to perform optimization with a single objective (or in the case of Abaqus; a weighted average of the specified objectives), multiple simulations have been performed with different volume constraints, ranging from 51% to 119% of the reference volume. The upper graph shows the displacement both in LC1 and LC2 direction, normalized wrt. the displacement of the reference geometry. The lower graph displays the same properties in the LC2 load case, but with strain energy as a measure of the structural stiffness, also showing the iterations during each optimization run. All curves display the boundary between feasible and infeasible solutions; hence the line represents the best possible result given the chosen constraints. This gives the designer knowledge of how changing one parameter will change another, and facilitates a reflected choice of final design.

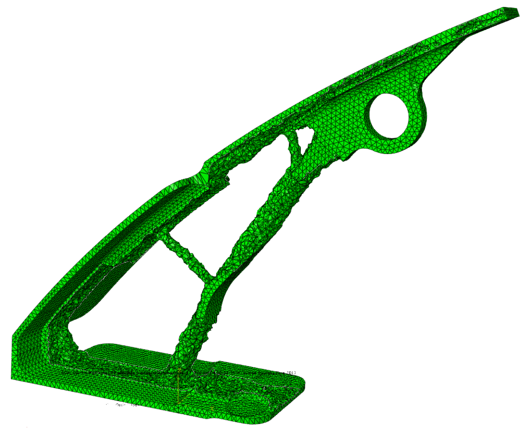
TV = Target Volume
DS = Design Space
RV = Reference Volume



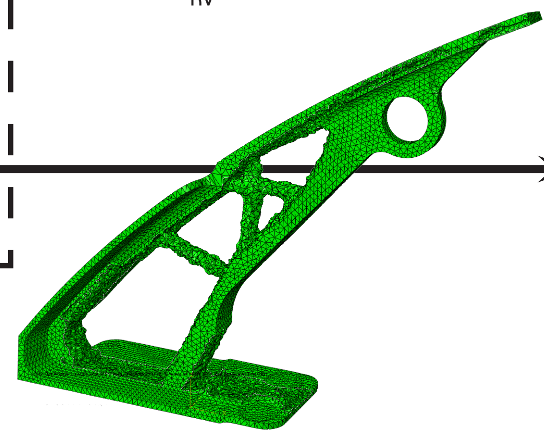




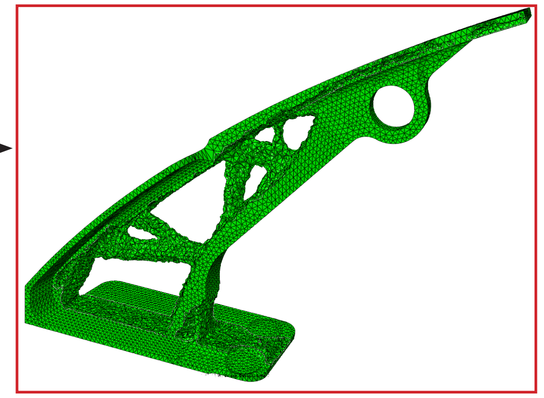
TV_{DS}=30%
TV_{RV}=51%



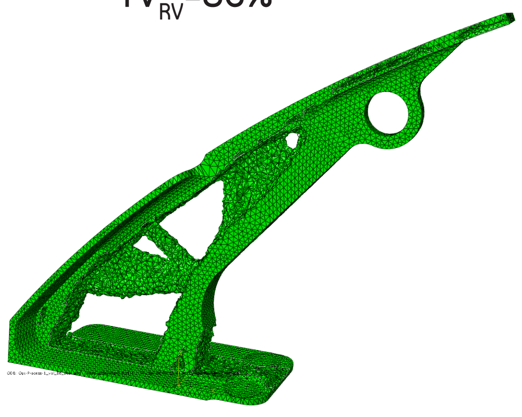
TV_{DS}=40%
TV_{RV}=68%



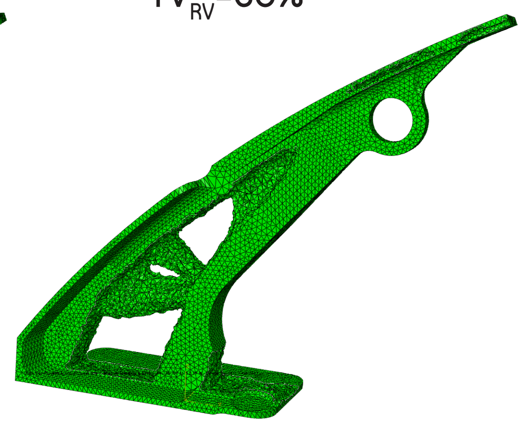
TV_{DS}=50%
TV_{RV}=85%



TV_{DS}=53%
TV_{RV}=90%



TV_{DS}=60%
TV_{RV}=102%



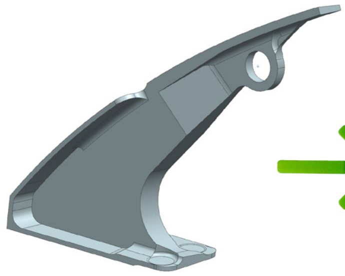
TV_{DS}=70%
TV_{RV}=119%

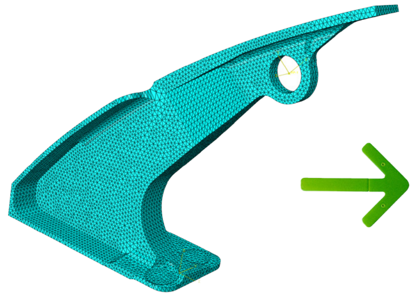
TOPOLOGY OPTIMIZATION process description using Abaqus ATOM

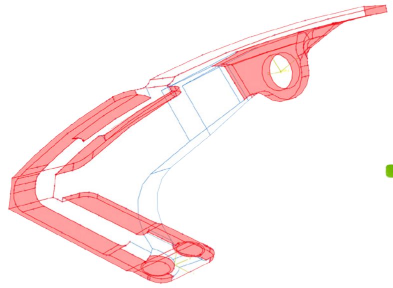
Topic: Topology optimization; minimize volume and maintain displacement, summary | Name: Steffen Johnsen | Date: 15.04.2013 | Approved by: Terje Rølvåg

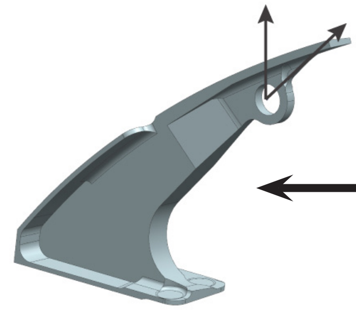
1. Setup a normal static analysis containing all boundary conditions. Both linear analysis and geometric nonlinear analysis are supported.
2. Evoke the Optimization environment, and create a new Optimization task, and define a name for the optimization. Make sure to choose whether to use the General algorithm or the Stiffness algorithm.
3. Define all Design responses in the Design Response Manager (all properties that will be influenced in the analysis; volume, eigenfrequencies, strain energy, displacement, etc.)
4. Choose which Design response(s) to use as an Objective in the Objective Function Manager. Weigh multiple objectives against each other and choose whether to maximize, minimize or minimize the max (preferred).
5. Define scalar constraints for the remaining Design responses.
6. Define Geometric Constraints (symmetry, frozen areas, cast condition, etc.). Use as few as possible, and run the first optimization without to see what is demanded.
7. Create an Optimization process in the Job Manager and specify which topology optimization to run. Start the optimization process.
8. Monitor progress by clicking Monitor during the optimization process in the Job Manager
9. When optimization has converged: Look at results by clicking Results in the Job Manager.
10. In the Job Manager: Click Extract and extract an STL- file from the final result. Make a copy of the .prt- file with the design space, import the STL file and use modeling tools and transform the part to look like the optimization result, or import the STL into an empty part and build the new part from scratch.


Original geometry

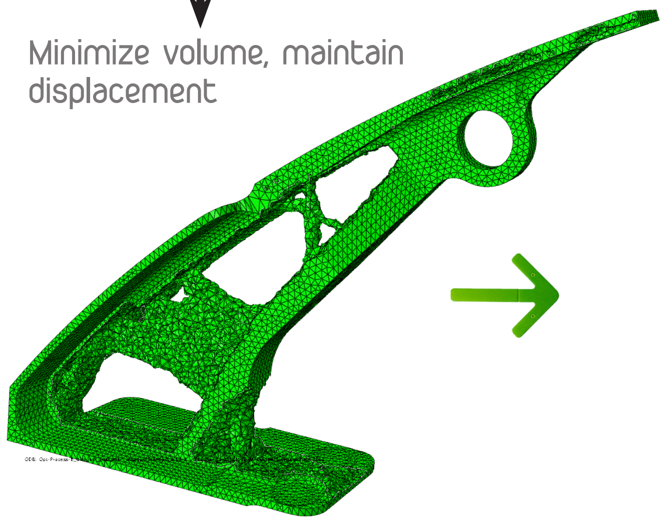

Design space

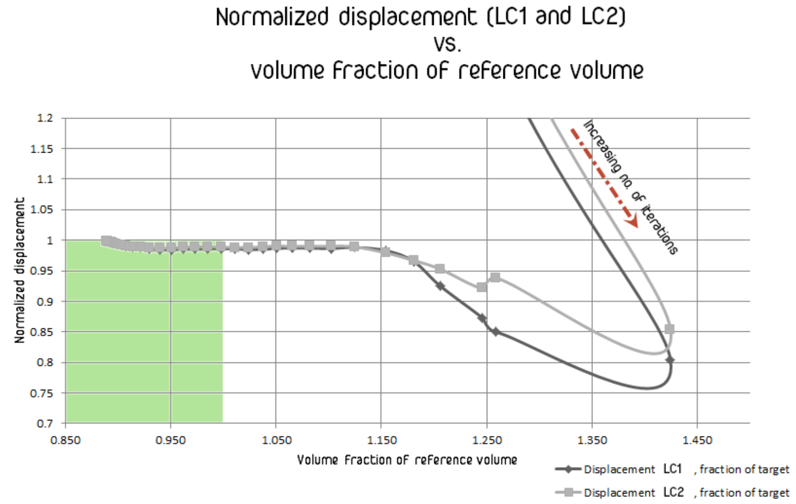

Meshed model

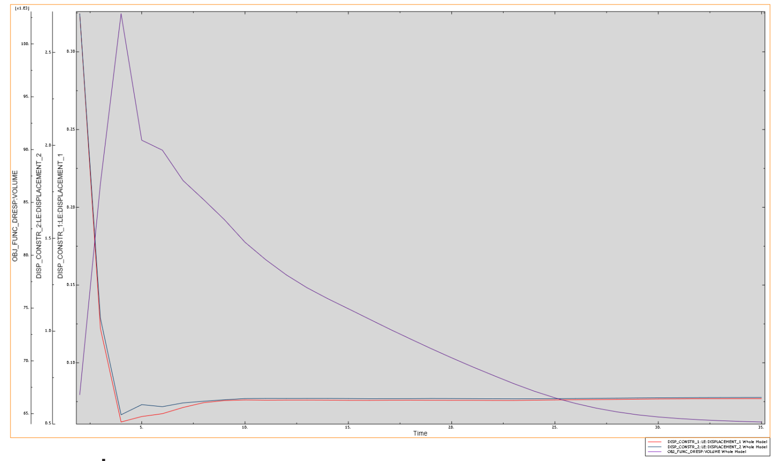

Model w/ geometric constraints

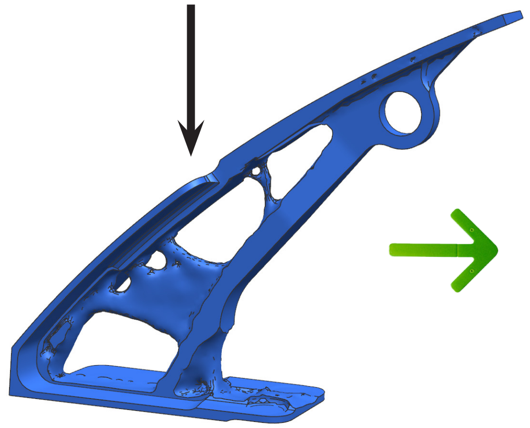

Loaded model (2.5 kN tensile and bending load defined in subsequent steps). Constrained at the bottom surface.

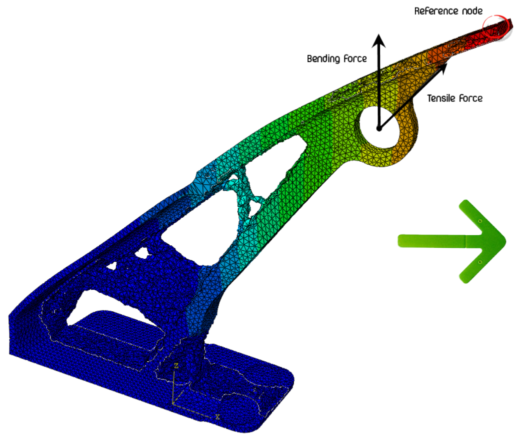
Minimize volume, maintain displacement


Optimization result

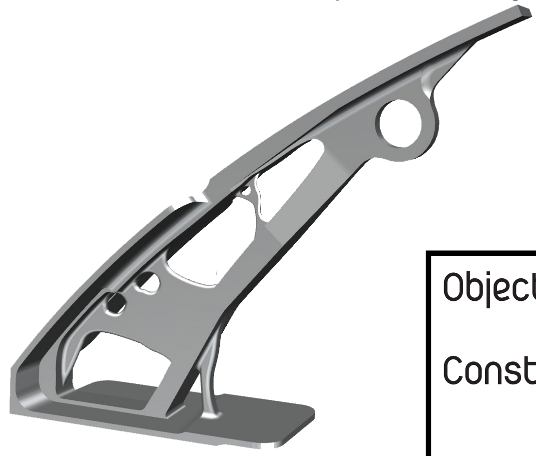



Optimization graphs


Extracted geometry



Results read- out


Reengineered part

Objective:	Minimize volume
Constraints:	Maintain displacement in load direc tions Delta_LC1 = 0.077 mm Delta_LC2= 0.644 mm
Geometric constraints:	Frozen areas (see figure), symmetry (not for frozen areas) and forging (no inner voids).

WEIGHT SAVED: 11.2% (OPTIMIZATION RESULT)
7.11% (REENGINEERED RESULT)

CHANGE IN STIFFNESS (REENGINEERED RESULT):
LC1 DIR.: + 3.90%
LC2 DIR.: + 0.62%

suplight

UGS The PLM Company | modeFRONTIER | NTNU

TOPOLOGY OPTIMIZATION process description using Abaqus ATOM

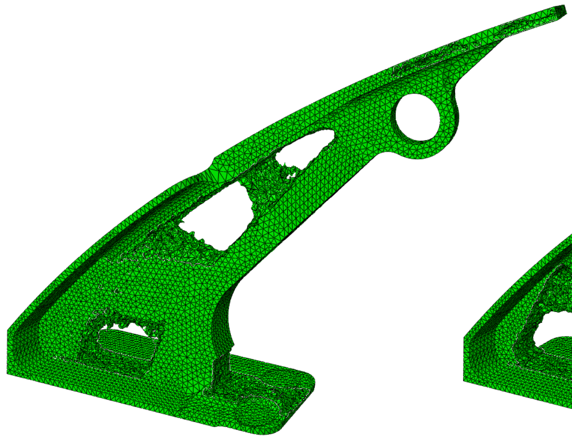
Topic: Topology optimization; minimize volume and maintain displacement in force direction | Name: Steffen Johnsen | Date:05.04.2013 | Approved by: Terje Rølvåg

Another method when performing topology optimization is based on MINIMIZING VOLUME (objective) and PRESCRIBING THE DISPLACEMENT. The solution method then provides a solution with a PRESCRIBED STIFFNESS (if the solution converges) and the LOWEST POSSIBLE VOLUME, given the design space and constraints. These constraints can be specified for the overall displacement magnitude of a load case or for specific directions. This sheet shows the HAI hinge optimized with this method and similar geometric constraints as specified earlier in the brief.

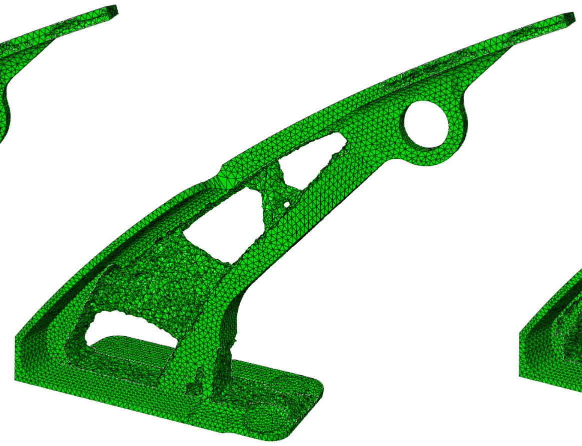
Constraints:
$Delta_{Tensile}$ = 0.077 mm
$Delta_{Bending}$ = 0.644 mm

The graph shows the resulting displacement and corresponding volume, normalized with regards to the reference geometry (the original part). Nodes in the graph represents iterations in a single optimization run.

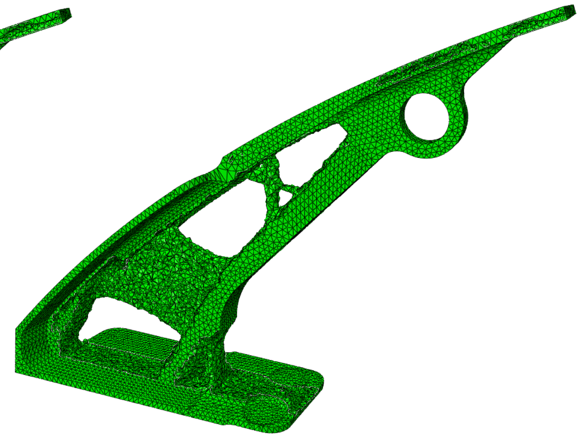
Results show that the stiffness is maintained, and that there is a net weight saving of 11.2%. The gross weight saving after interpreting the results into a solution that is easier to manufacture will probably be closer to 10%, still fulfilling the initial target weight saving.



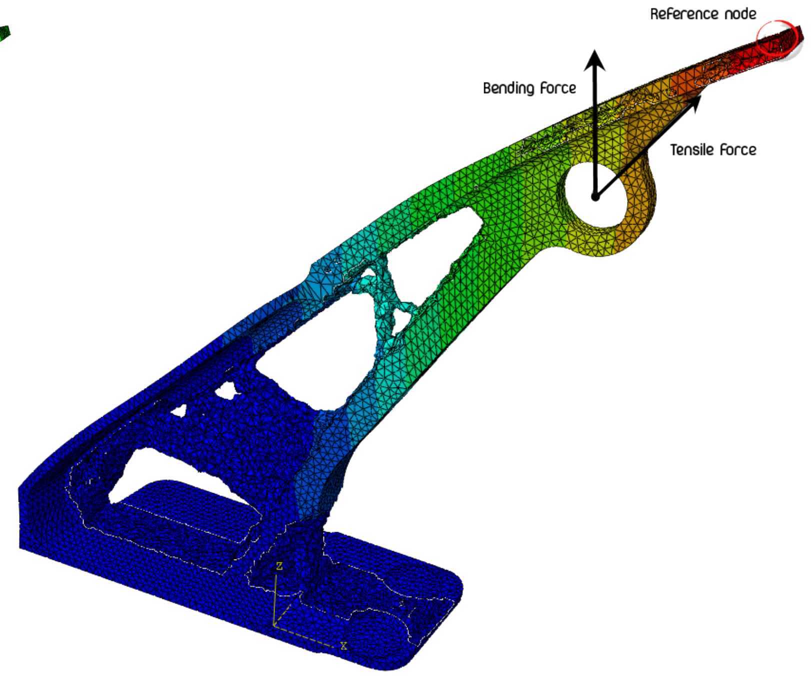
Iteration no. 10



Iteration no. 18



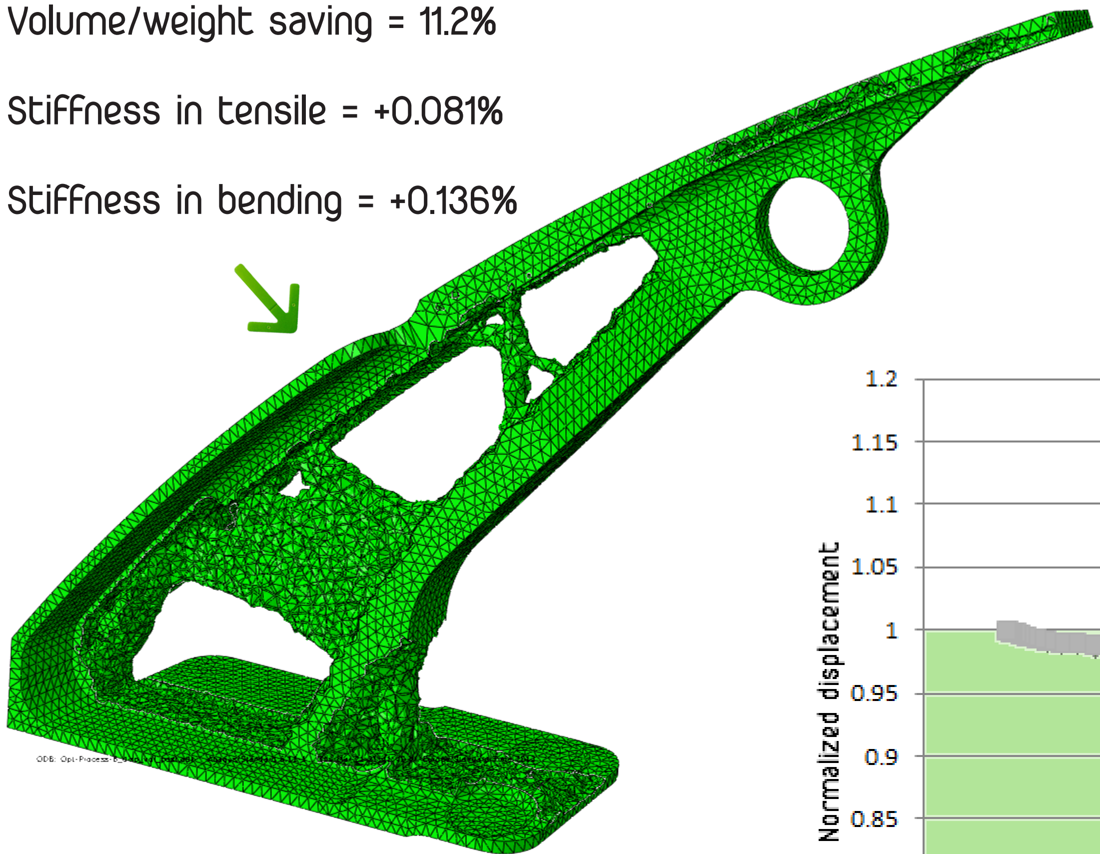
Iteration no. 15



Volume/weight saving = 11.2%

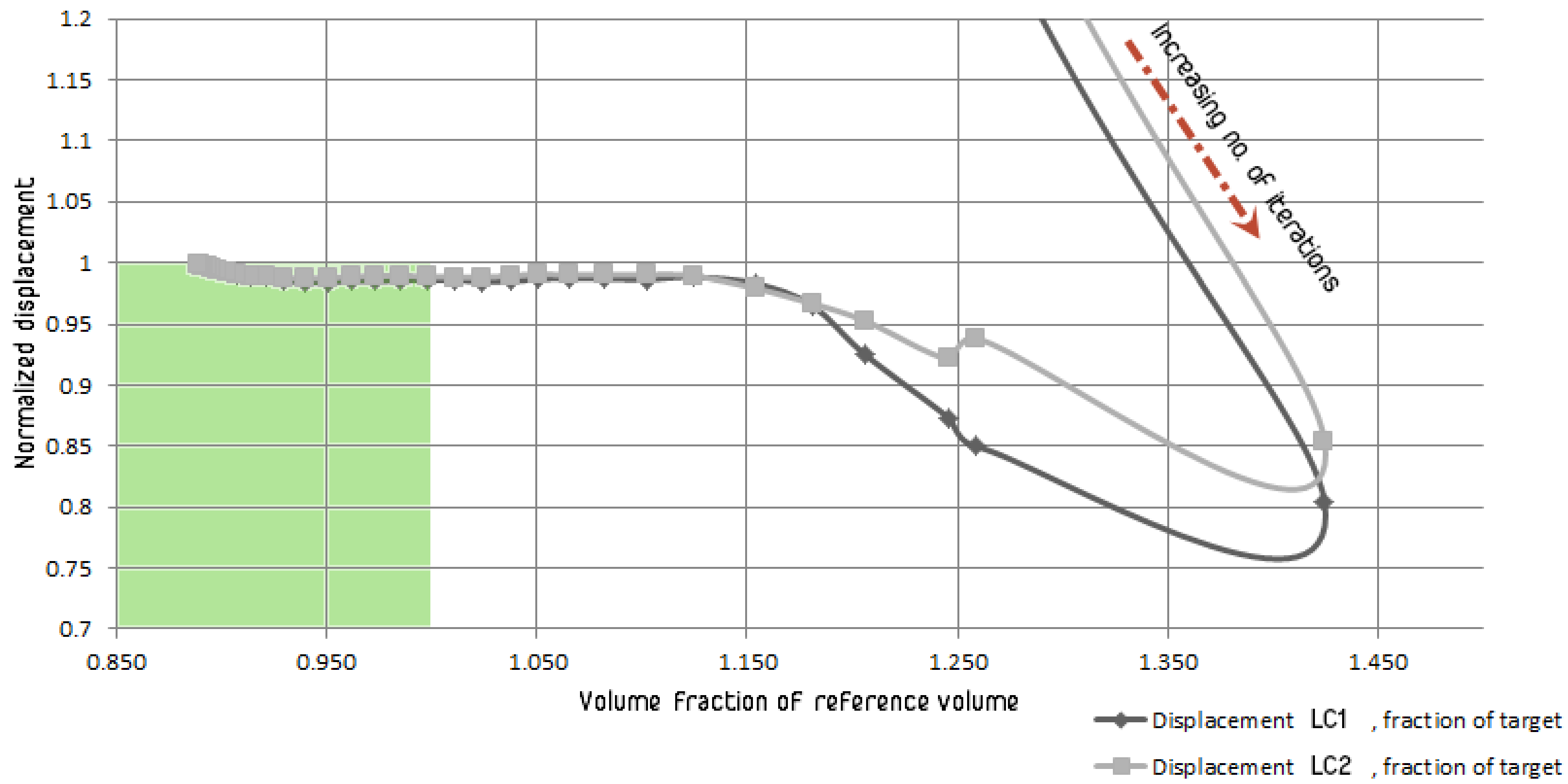
Stiffness in tensile = +0.081%

Stiffness in bending = +0.136%



Iteration no. 32 (last)

Normalized displacement (LC1 and LC2) vs. volume fraction of reference volume



suplight

UGS The PLM Company | mode FRONTIER | NTNU

TOPOLOGY OPTIMIZATION process description using Abaqus ATOM

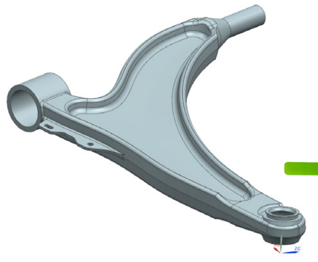
Topic: Topology optimization; minimize volume and maintain displacement, summary

Name: Steffen Johnsen

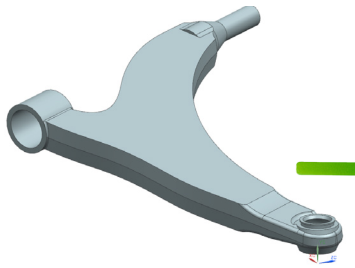
Date: 15.04.2013

Approved by: Terje Rølvåg

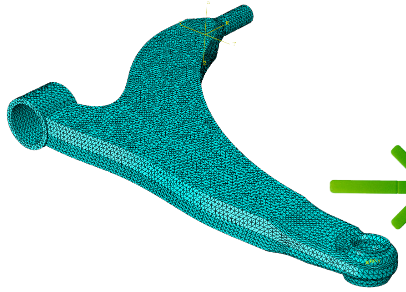
1. Import model and partition geometry into regions. This will make it easier to define frozen areas later in the analysis.
2. Setup a normal static analysis containing all boundary conditions. Both linear analysis and geometric nonlinear analysis are supported.
3. Evoke the Optimization environment, and create a new Optimization task, and define a name for the optimization. Make sure to choose whether to use the General algorithm or the Stiffness algorithm.
4. Define all Design responses in the Design Response Manager (all properties that will be influenced in the analysis; volume, eigenfrequencies, strain energy, displacement, etc.)
5. Choose which Design response(s) to use as an Objective in the Objective Function Manager. Weigh multiple objectives against each other and choose whether to maximize, minimize or minimize the max (preferred).
6. Define scalar constraints for the remaining Design responses.
7. Define Geometric Constraints (symmetry, frozen areas, cast condition, etc.). Use as few as possible, and run the first optimization without to see what is demanded. For the FLCA, whole regions should be set to frozen.
8. Create an Optimization process in the Job Manager and specify which topology optimization to run. Start the optimization process.
9. Monitor progress by clicking Monitor during the optimization process in the Job Manager
10. When optimization has converged: Look at results by clicking Results in the Job Manager.
11. In the Job Manager: Click Extract and extract an STL- file from the final result. Make a copy of the .prt- file with the design space, import the STL file and use modeling tools and transform the part to look like the optimization result, or import the STL into an empty part and build the new part from scratch.



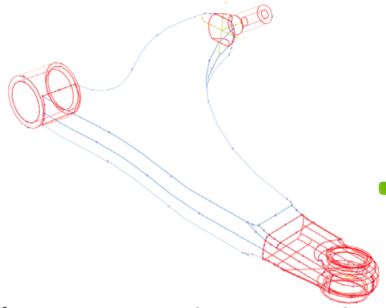
Original geometry



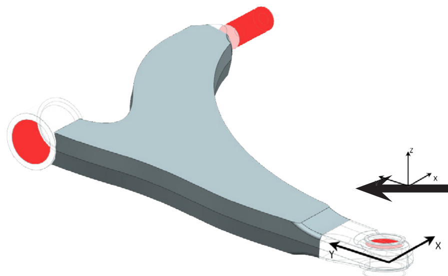
Design space



Meshed model

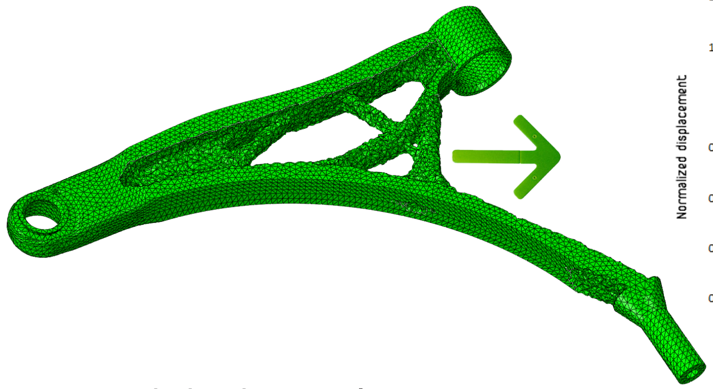


Model w/ geometric constraints



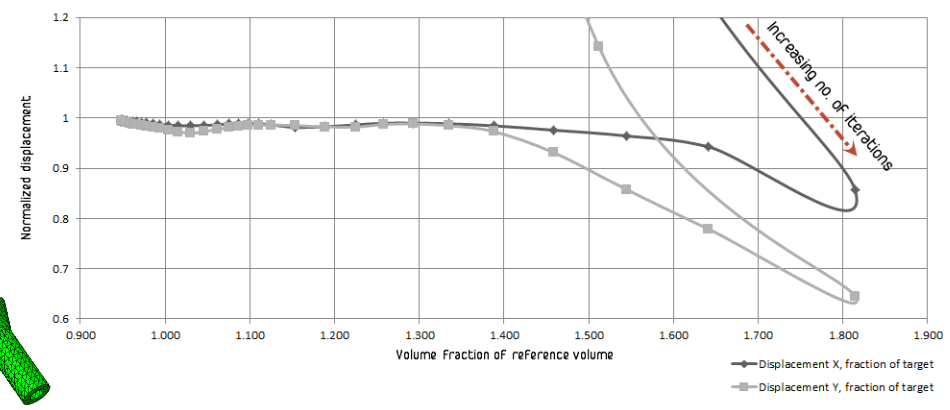
Loaded model (7 kN in X- and Y- direction defined in subsequent steps). Constrained at the bottom surface.

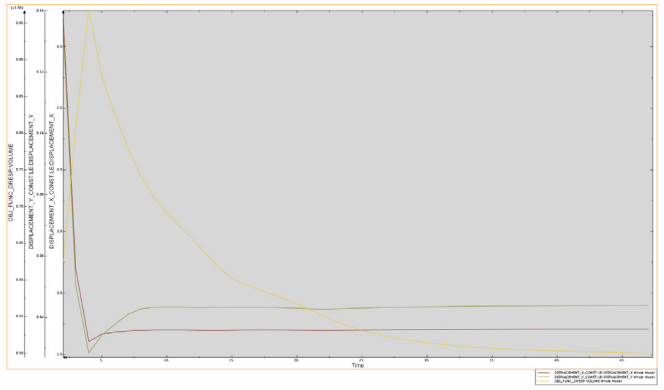
Minimize volume, maintain displacement



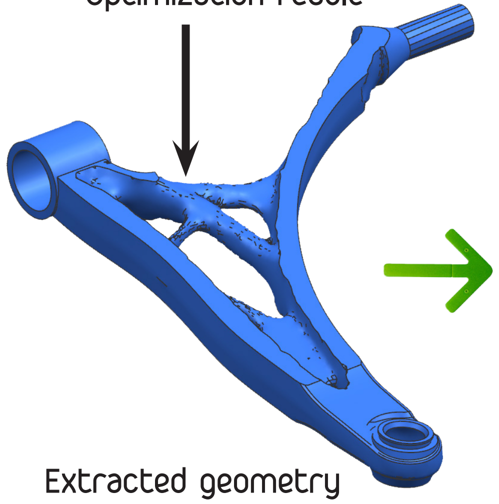
Optimization result

Normalized displacement (X- and Y- direction)
vs.
volume fraction of reference volume

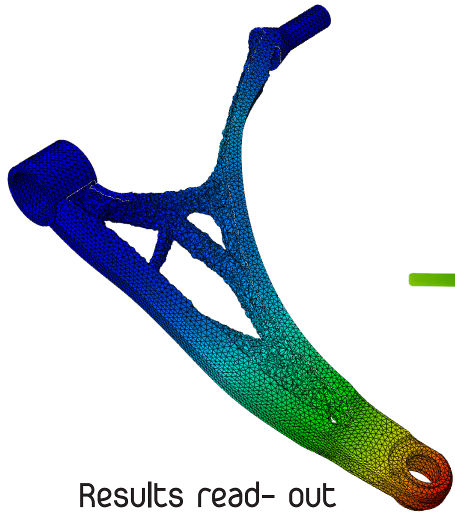




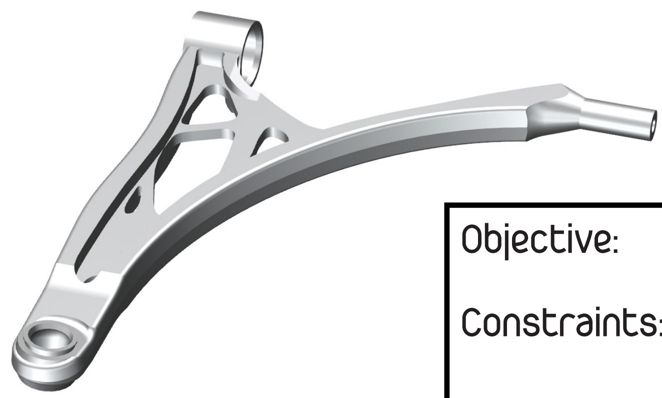
Optimization graphs



Extracted geometry



Results read- out



Reengineered part (illustrational picture)

Objective:	Minimize volume
Constraints:	Maintain displacement in load direc tions Delta_X = 1.418 mm Delta_Y = 0.044 mm
Geometric constraints:	Frozen areas (see figure), symmetry (not for frozen areas) and forging (no inner voids).

WEIGHT SAVED: 6.41% (OPTIMIZATION RESULT)
5.92% (REENGINEERED RESULT)

CHANGE IN PROPERTIES (REENGINEERED)
DISP. X-DIR.: 0%
DISP. Y-DIR.: 0%
BUCKLING LOAD: +9.88%

suplight

TOPOLOGY OPTIMIZATION process description using Abaqus ATOM

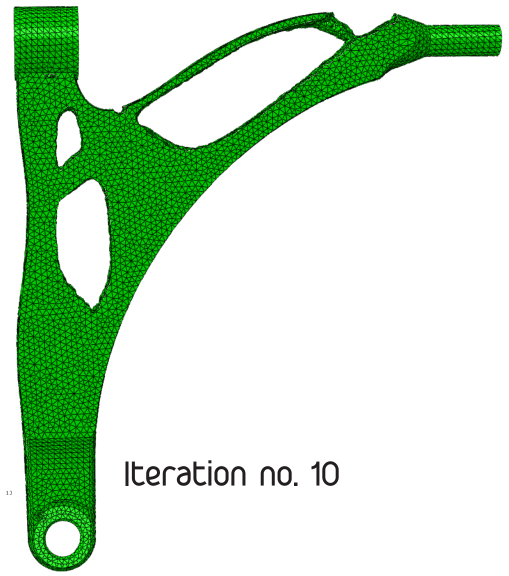
Topic: Topology optimization; minimize volume and maintain displacement in force direction | Name: Steffen Johnsen | Date:05.04.2013 | Approved by: Terje Rølvåg

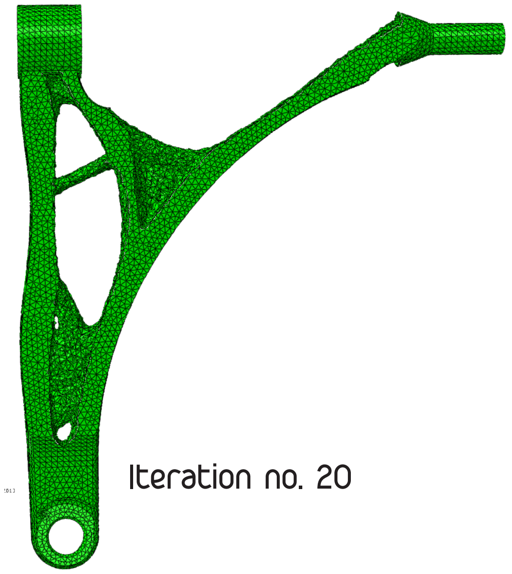
The method used when performing topology optimization is based on MINIMIZING VOLUME (objective) and PRESCRIBING THE DISPLACEMENT. The solution method then provides a solution with a PRESCRIBED STIFFNESS (if the solution converges) and the LOWEST POSSIBLE VOLUME, given the design space and constraints. These constraints can be specified for the overall displacement magnitude of a load case or for specific directions. This sheet shows the Front Lower Control Arm optimized with this method and similar geometric constraints as described on the previous page.

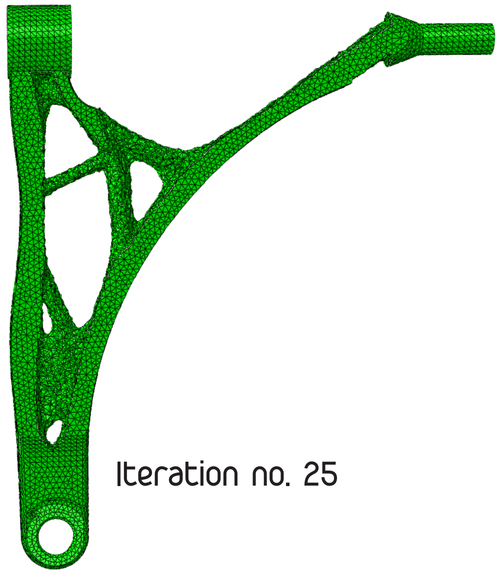
Constraints:
Delta$_X$= 1.418 mm
Delta$_Y$= 0.044 mm

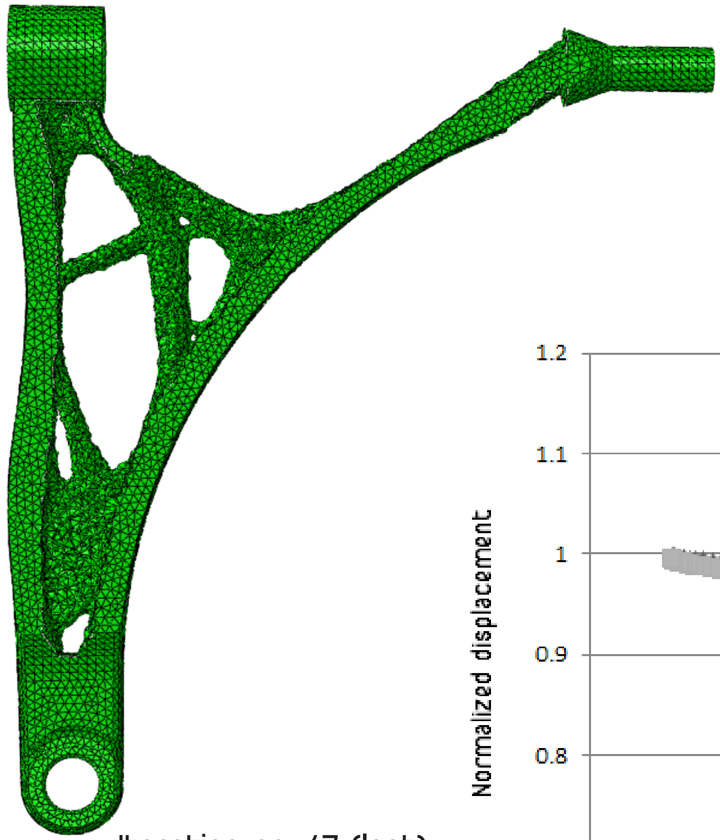
The graph shows the resulting displacement and corresponding volume, normalized with regards to the reference geometry (the original part). Nodes in the graph represent iterations in a single optimization run.

Results show that the stiffness is maintained, and that there is a net weight saving of 6.41%. The gross weight saving after interpreting the results into a solution that is easier to manufacture will probably be closer to 5%, still fulfilling the initial target weight saving.


Iteration no. 10


Iteration no. 20


Iteration no. 25

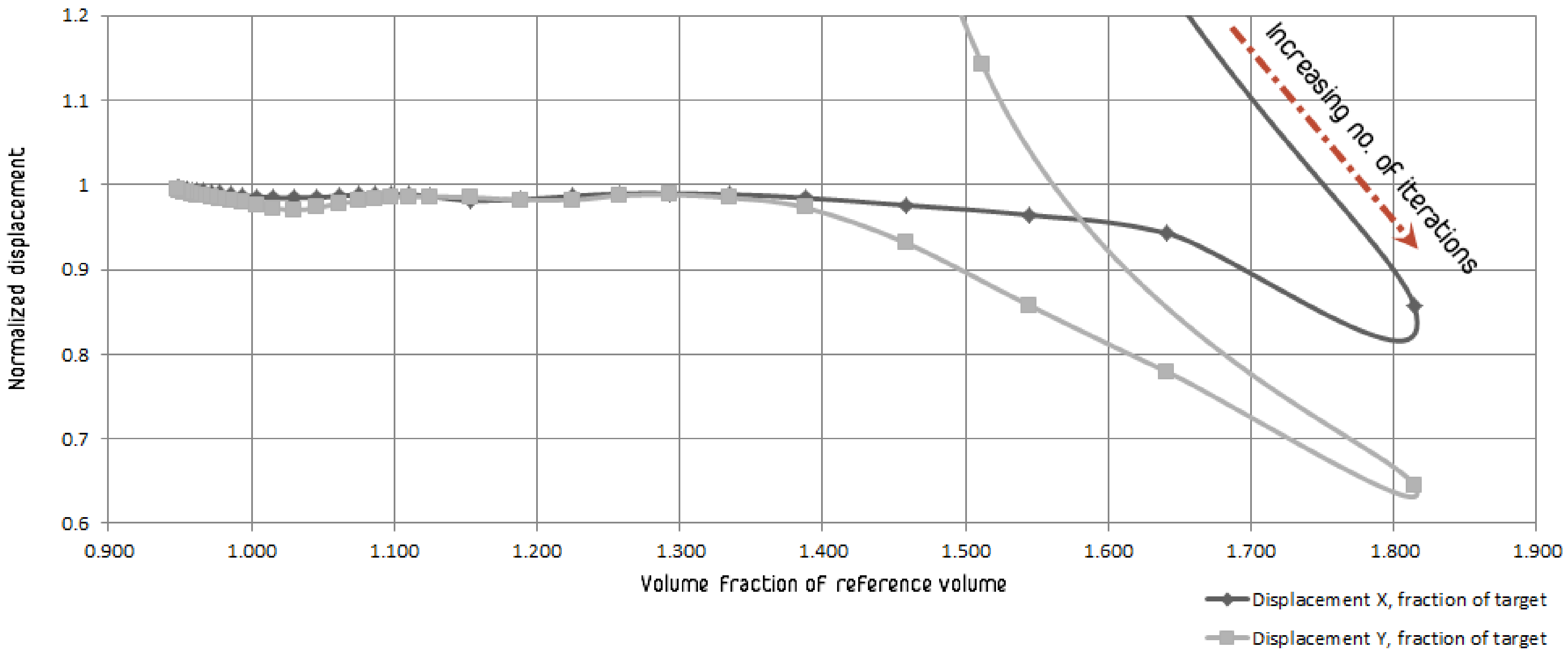

Iteration no. 47 (last)

Volume/weight saving = 6.41%

Stiffness in X-direction = +0.110%

Stiffness in Y-direction = +0.157%

Normalized displacement (X- and Y- direction)
vs.
volume fraction of reference volume



suplight