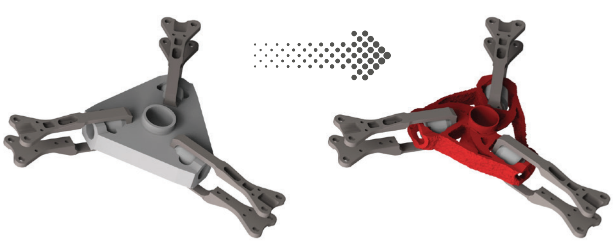
ANSYS Topology Optimization

Topology Optimization is the generation of a shape which fulfils some criteria by an automated mechanism. It is often used to find out the minimum amount of material need to fulfil some load case or design goals.

Optimization as a whole seeks to remove the human limitations inherent in reaching some ideal design. Parametric optimization does this by allowing software to vary design parameters such as thickness, radius size, number of holes or webs, etc., on a part and finding the combination that best meets some goal. Automation enables the software to choose a sub-set of all the permutations and decide on the best combination to reach the goal.

Where topology optimization differs from traditional parametric optimization is that it removes the boundaries imposed by varying only parameters, to the exclusion of other possible variations. The shape that is produced is not constrained by the original geometry other than the space it occupies.



Introduction

Topology optimization is a solver-based approach to optimization. The analysis has a load case, or load cases, that describe the physics that the part (or assembly) undergoes; it then uses this information to remove material from an initial design. A simple example might be a support bracket with a push load and a sideways load.

The optimization uses the mesh of the part as the input. Doing so means that the solver can calculate the requirement for each element within the model and the impact it has on the overall solution. Elements are “removed” from the optimization if they do not contribute to the final solution. The end result of an optimization is a shape that is somewhat dependant on the starting mesh.

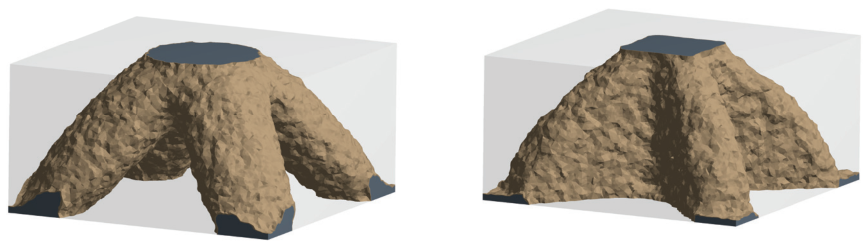
Analysis controls

If the model is meshed with a very coarse mesh then the problem will of course run very quickly, but the resulting shape can only be as good as the starting mesh. Increasing the mesh density will result in a more refined result, but the calculation will take longer to perform.

A response constraint determines what the topology optimization is trying to achieve. The most common use case is a mass constraint. The solver here is trying to find a shape which uses only a certain percentage of the original part’s mass. Constraints to maintain certain stress limits on a whole part or in specific areas can also be used, as well as those that limit displacements or reaction forces induced by loadings on the structure.

It's also sometimes useful to look at natural frequencies. Removal of material will significantly alter the way a structure vibrates. The combination of a structural load case and a vibration load case could also be valuable.

One of the main drivers for carrying out topology optimization is to take advantage of the freedom that additive manufacture brings. Often, though, companies may want to use topology optimization for traditional manufacturing. This means that some constraints need to be added to the analysis in order to produce a shape which can be manufactured.There may also be a need for controls to ensure that fixing points or mounting points are maintained.



Adding manufacturing constraints. Model on right has been optimized for casting

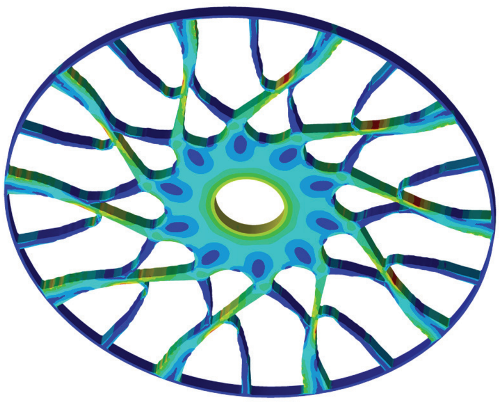
Topology optimization controls could be as simple as reserving certain geometry features as no-go areas for material removal or specifying minimum material thickness.

Minimum wall thicknesses are also important for manufacturing. Certain processes — like casting and, to an extent, machining — require that feature sizes on parts do not fall below certain values. For casting this could be to ensure that molds are properly filled during the casting process. Very fine features could cause issues that lead to an area that is ill-cast, and that features elsewhere in the part are not properly made.Controls like this can help to ensure that parts are actually manufacturable and that the whole process of going through topology optimization is a worthwhile one.

Depending upon the type of part, it may be desirable to ensure that features of the resulting part can be machined only from a particular direction. For example a 3- or 4-axis cnc machine may only have a limited range of motion. By controlling the way the features like ribs or holes are generated on the part, the machine manufacturing the part can complete the process and the goals of the optimization can still be met.

For rotating parts, or parts that have equal loadings in multiple directions, it can be helpful to also specify either planar or cyclic symmetry. Planar symmetry could be as simple as ensuring that the 4 legs of a support structure are even, or the requirement could be more complex , such as a case in which the loading is not equal but the design needs to be symmetric for weight distribution or merely aesthetics.

For cyclic symmetry it is often important to produce a design that has an even distribution of weight around an axis. Failure to do so would have obvious consequences once the part begins to rotate and the unbalanced part causes vibrations. Specifying cyclic symmetry and the number of cyclic sectors will allow the software to produce a repeating pattern about an axis, rather like an alloy wheel on a car or gear wheel.

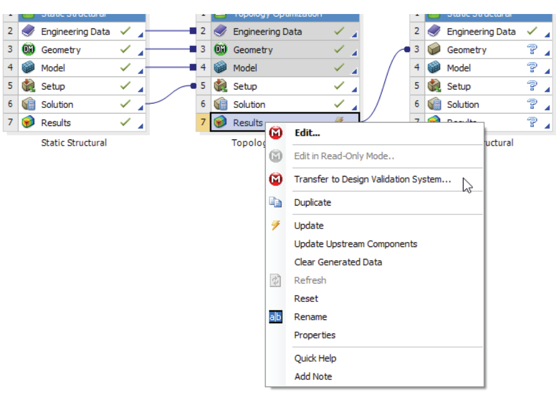


Optimization using cyclic symmetry showing stress plot of loaded validation model

Geometry preparation and model validation

The last challenge of the topology optimization process is a significant one. Because, as already discussed, the solution takes place on the mesh, the result is not a clean piece of geometry, but is instead a faceted mesh. In order to have the freedom to generate an arbitrary shape, the solver needs to use a mathematical approach based upon the contribution each element within the model makes to the solution.

The output from the solver is often organic looking and, depending on the mesh size, quite irregular in its surface appearance. In order to make use of this shape, further work must be done. Options to export this shape vary, but ANSYS Mechanical is able to export an STL file of the part or assembly. This file can then be cleaned up and made ready for validation. The clean-up will need to ensure that required features such as bolting holes or mating surfaces, where the part interacts with other parts in the assembly, meet the design needs, and that any jagged edges or holes are smoothed over.



Inserting a Design Validation System

ANSYS SpaceClaim can do all of this and can often automate certain steps to improve the user experience. Smoothing and wrapping tools can turn a laborious process into one that doesn't take too long. The original geometry of the un-optimized part can be used as a framework for holes and reference surfaces, further accelerating the process of geometry preparation.

Once the geometry has been prepared and turned from a faceted surface model (STL file) into a solid piece of CAD, it can be validated.Validation of the optimized topology is vital to ensure that any changes made in the geometry edits or the addition of other loading scenarios do not invalidate the design. The ANSYS Workbench framework provides a direct link between the topology optimization analysis and a validation model. The same loads present in the upstream analysis are available to perform the validation step.

Once model validation is complete and the results are checked to ensure that the resulting design meets requirements, the optimization is complete. The process of carrying out topology optimization is very efficient and can result in significant improvements in part performance. There are a lot of tools available to control exactly how the analysis takes place.

ANSYS, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where ANSYS software played a critical role in its creation. ANSYS is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination. Visit **www.ansys.com** for more information.

© 2017 ANSYS, Inc. All Rights Reserved.