

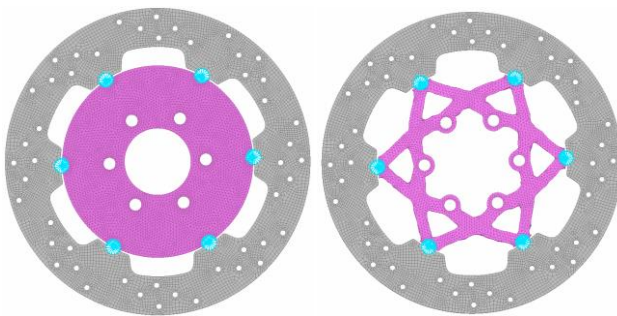
Topology and Shape optimization within the ANSA-TOSCA Environment

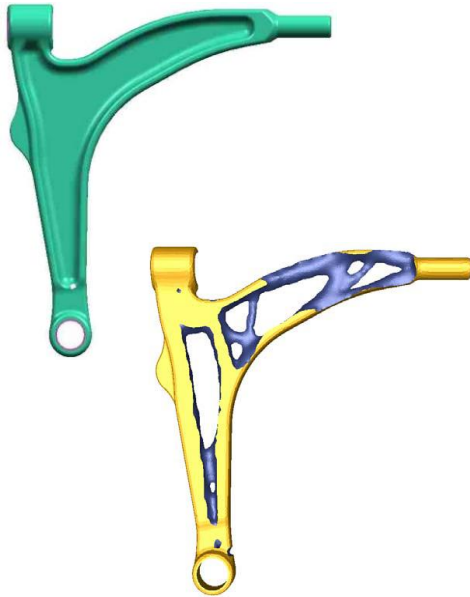
Introduction

Nowadays, manufacturers need to design and produce, reliable but still light weighting and elegant components, at minimum cost. Fortunately, the computational resources are continuously growing, offering new options for optimum designing, within reasonable time periods. Due to the remarkable reduction in the development time of simulation analysis, it is possible to quickly and precisely define the necessary modifications needed of a part's shape. The final aim is the reduction of the total mass while keeping the stresses at acceptable level. **TOSCA structure** in conjunction with the **ANSA** powerful **pre-processing** package, offers a really innovative tool for such applications. The established **TOSCA** Optimizer takes advantage of the **ANSA's** powerful pre-processing functionality and user-friendly interface, in order to effectively apply Topology and Shape Optimization algorithms. This paper focuses on the key elements of the TOSCA – ANSA Environment [TAE] that presents a reliable know-how in the design - manufacturing process.

Problem definition

The definition of the optimum shape for a completely new or improved component that will fulfill mechanical standards and manufacturing or operational constraints is a common requirement. The objective is to reach to a shape with the optimum weight while at the same time respect any restrictions in the areas of assembling with neighboring parts, and the application of boundary conditions. Simultaneously, it is also usually needed to provide the model with some extra shape characteristics, such as symmetry, minimum member size etc.





Process description

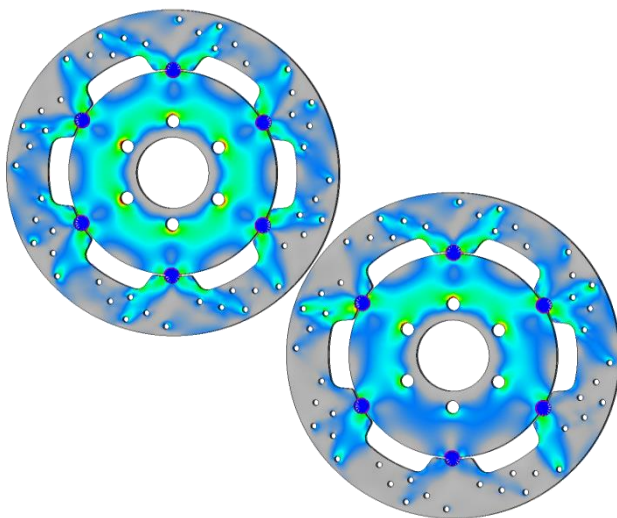
A common approach in modern designing is to start with the initial solid model. The next step is the definition of the design area, which is the maximum allowed area for the component, by excluding some areas of the model that should not be affected by the topology optimization. Such areas can be those of necessary features for the assembly or areas of boundary conditions.

Additionally, a variety of manufacturing conditions are directly defined in TAe. These will guide the algorithm to produce feasible for manufacturing design shapes. Additionally, other geometrical or operational Constraints can also be defined.

In the first step of the process, a **Topology** Optimization is applied. Here, material is gradually removed from the model, in a number of iterations. The algorithm determines a new material distribution by removing elements from the Design Area.

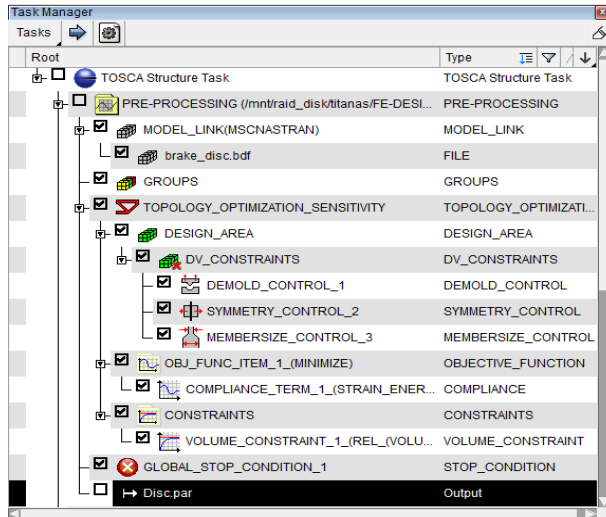
After the Topology Optimization, the general Shape of the component is more or less fixed. In the second phase, a **Shape** Optimization is applied. Here selected nodes are moving, slightly changing the shape. The target is to further minimize and equalize the stress concentrations.

Case Presentation



This case demonstrates the Topology and Shape optimization process for a drilled Brake Disc. The inner disc that connects the wheel with the outer Disc (braking surface), is the area that will be optimized (Design Area). Simple Point Constraints are applied to the holes that fit the Disc to the wheel, defining these areas as fixed. Two load cases are set. A moment is applied to the breaking Disc, and a tangent Force at the area where the breaking pads push the Disc during braking. These loads are easily and quickly defined in ANSA.

As it is shown, after the first run, high stresses are concentrated around the



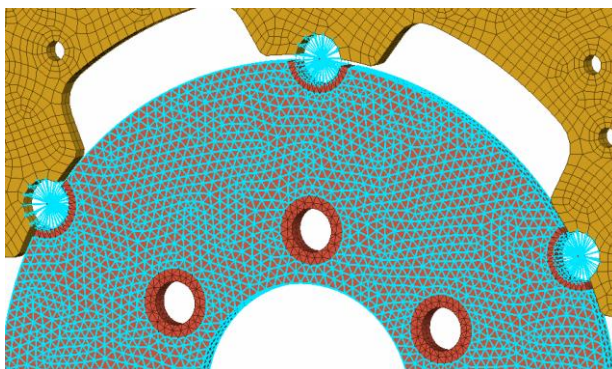
constrained areas. In contrast, there are extended areas inside the Design Area that are almost under no stress. This offers enough space and freedom for the Topology Optimization Algorithm to remove material during the step by step procedure.

The ANSA – TOSCA Environment

The whole process is consolidated in **TOSCA-ANSA** Environment [Tae]. The ANSA tool that represents the TOSCA Structure is the Task Manager. This is a general ANSA Tool that can organize and automate the process of user-defined, pre-processing Tasks.

In the Task Manager, special pre-defined tasks have been created for the Topology, Shape, and Bead Optimization of the TOSCA Structure. The users can define all the necessary TOSCA keywords interactively and quickly, in a step wise sequence, taking advantage of the extended Task manager functionality and the user friendly interface. Additionally, some necessary ANSA algorithms, such as reconstruct and volume meshing, are embedded actions in the Tosca Task. The calling of TOSCA as well as the Solver is also done directly through the Task Manager.

Topology Optimization



Design Area – Topology Optimization

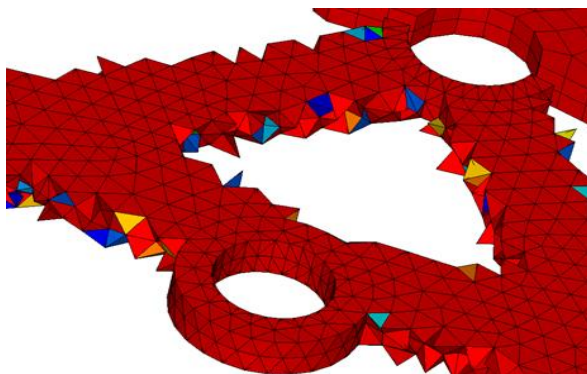
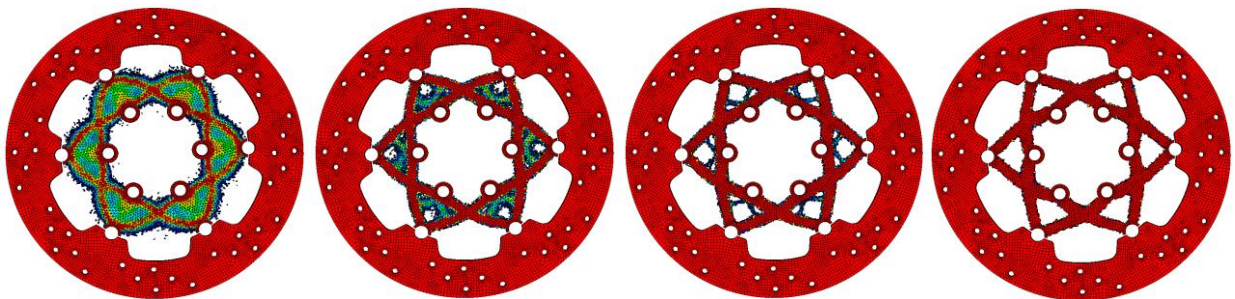
At the beginning, the initial (full solid) model, saved in solver format, is input in ANSA through the TOSCA Task. The first step is to define the **Design Area** of the model. The Design Area is the region of the model where the TOSCA Topology algorithm is permitted to remove material. Using the ANSA functionality for the definition of **Sets**, this area is interactively and quickly defined. Usually, some element zones, around the crucial features of the component or near to the Boundary Conditions, are excluded from the Design area, to ensure that they will remain intact.



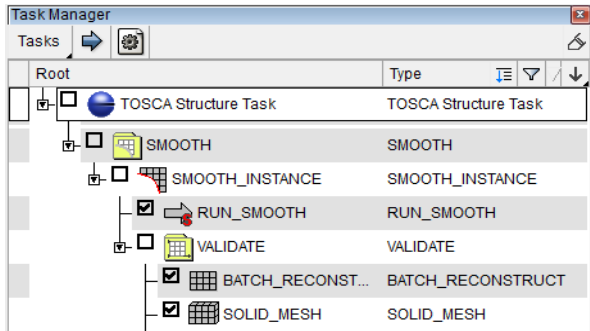
In similar to the brake disc cases, a Symmetry Constraint is also necessary. The symmetry here needs to be defined as rotational, at a step of 60 degrees. Other possible Constraints might be a member-size control or a Frozen Area constraint.

In the next step, the Objective function is defined. There is a wide variety of available functions depending on the results of the FE analysis (Design Responses). The minimization of Compliance (which is actually the maximization of stiffness), is one of the most common. Finally, Constraints for the problem should be defined, based again on the design Responses. Such Constraints can be the total volume, weight, displacements etc. In this example the final volume was set to 40% of the initial. The maximum number of iterations is defined and the output file for TOSCA (.par file) is automatically created by ANSA. The TOSCA algorithm is now ready to run. During each optimization cycle, TOSCA first removes elements and afterwards the solver is called in to calculate the results for the new design. The results are re-evaluated by TOSCA and a new iteration starts.

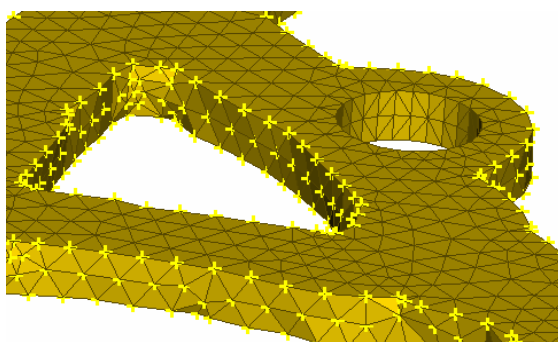
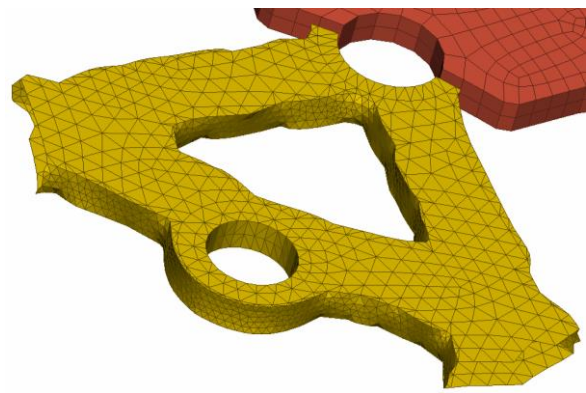
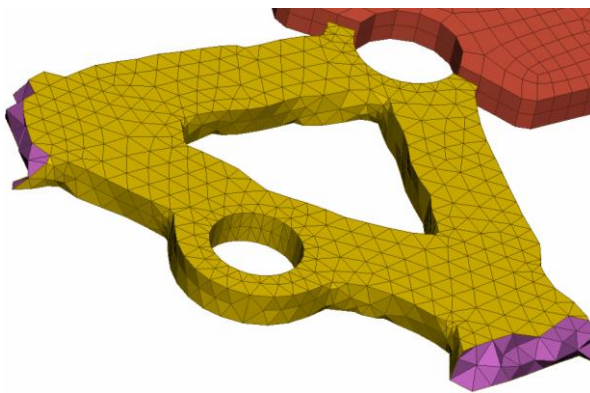
The time demanded for the completion of all iterations depends mostly on the time needed for the solver to run the model. In the Disc example less than an hour was enough to run all 38 iterations. Then, ANSA calls the TOSCA Interface which visualizes the intermediate steps of material removal. In this example, the application of the symmetrical symmetry constraint and the gradual material removal can be easily demonstrated.



All the intermediate models as well as the final one, have definitely abnormal surface. This is not a problem since the TOSCA **Smooth** Algorithm sweeps the model very quickly giving a smooth surface for the final design proposal. At this state, all the solids of the Design Area have been removed and the new shape is only described by a Shell Mesh. Inconsistencies in the Element length are automatically fixed by the powerful and fully customizable **Reconstruct** algorithm of ANSA. The algorithm is embedded under the



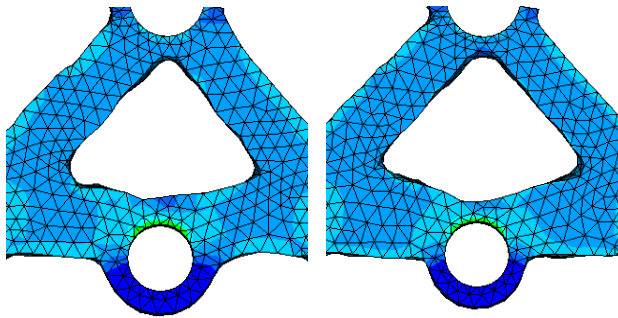
Smooth Item of the Tosca Task and is applied in one step. In the same way, the reconstructed shell mesh is **rapidly** automatically filled with tetra solid Mesh, by the incorporated Solid Mesh algorithm. After all these modifications, the Boundary conditions of the initial model are automatically translated and re-applied to the new design. Thus, the new model is ready to run again to be re-evaluated without any user intervention.



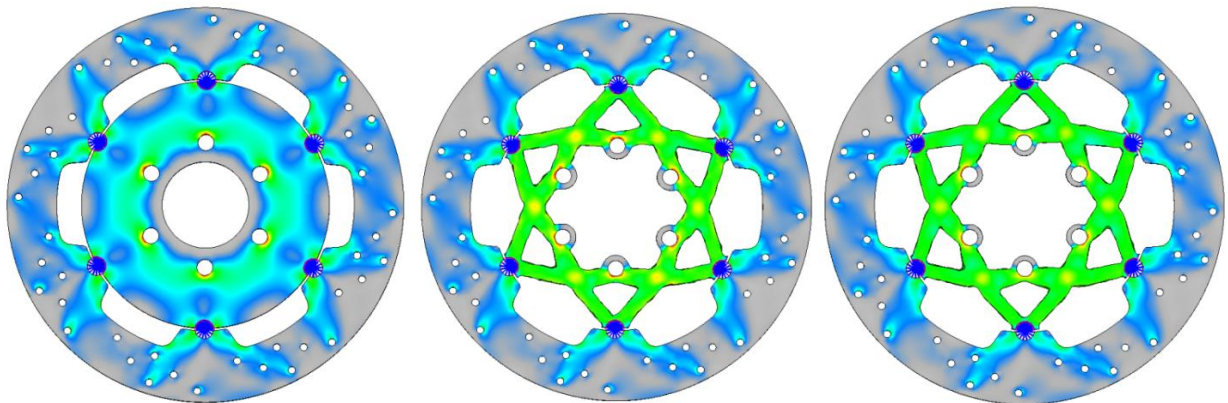
Design Area – Shape Optimization

Shape Optimization

After Topology Optimization, the general layout of the component is nearly fixed. Consequently, a Shape Optimization is applied in order to minimize stress concentrations. The Shape modification through this step is not so remarkable and the total volume is usually kept the same. However, stresses are normalized significantly. For this process, **ANSA** provides another special pre-defined Task where the user can set up all the necessary actions for the **TOSCA** Shape Algorithm. The first step is again to define the Design Area of the problem which is a group of Nodes that are going to be moved in order to improve the model's behavior. According to the needs of the modeling, completely or partially Frozen Areas can be defined. After some iteration, the final



model is generated. The steps of Smoothing, Reconstructing and Solid meshing are followed similarly to the Topology Optimization. Finally, the new model is solved revealing the improvement in the stress distribution.



Initial Shape

Topology

Shape

Conclusions

Using the automated process of TOSCA - ANSA Environment it is possible to reach a component's optimum design proposal in an impressive short period of time. Keeping only 40% of the initial volume, we managed to maintain the maximum stress more or less the same ($\pm 1\%$). The Topology algorithm recognized the unstressed areas and gradually removed material, efficiently and intelligently. The result is an elegant component that fulfills the shape and operating needs. As it is expected, this component is more evenly stressed. However, after the material reduction there were still some stress concentrations. For this reason a Shape Optimization follows. Within a few iterations, it manages to reduce stress concentrations and minimize the peaks by shortly moving the appropriate nodes. The whole process is quick and user friendly. The duration of this process (shown in table below) depends more on the time needed for solving since the Optimization algorithms run rapidly. It is more than certain that TAe is a powerful combination that forms an efficient and useful Tool in the designing process.

	Topology Optimization	Shape Optimization
pre-processing set up	< 30min	< 20min
optimization & solver run	< 90 min	< 50min