



School of Engineering
Brown University

ENGN 1750: Advanced Mechanics of Solids

ABAQUS TOSCA OPTIMIZATION TUTORIAL

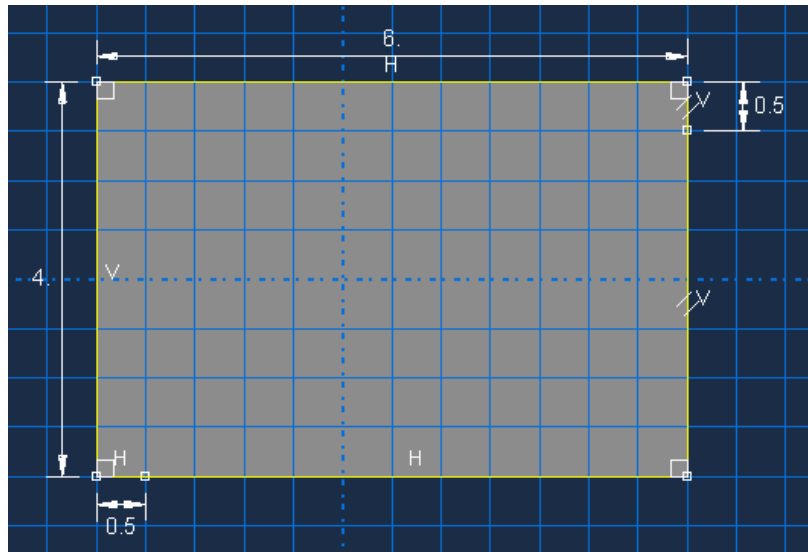
This tutorial will take you through all the steps needed to run the Tosca topology optimizer in ABAQUS/CAE

Background

When you design a part, you need to choose its shape. How do you know what the best shape will be? Usually we use our intuition to get a first approximation; then analyze its behavior, and maybe change the shape based on what we see in the predictions. But CAD software can sometimes find the optimal shape automatically. Topology optimization is still an active research field, and it is computationally intensive (especially if you need to optimize a part that needs to exhibit large deformations, or will deform plastically, like a crash rail). But it can still be cheaper to have a supercomputer run for a few weeks than to pay someone with a PhD to do the design calculations. ABAQUS/CAE has an add-on optimizer called 'Tosca' that can be used in this way.

Using Tosca

We will start by running a simple 2D (plane stress) structural optimization problem. Our goal is to find the optimal shape of a plate that will be clamped near one corner, and will support a load (specified as a traction) near another.

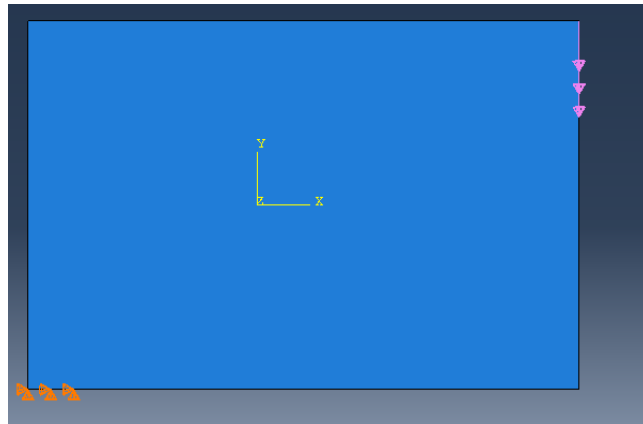


1. In the Part module, draw a rectangular shape. The dimensions are mm. Be sure to include the straight line segments near the bottom left and top right corners: these will be used to specify boundary conditions later.

2. In the Property module, create a material with Young's Modulus 210 GPa (use forces in N and length in mm), Poisson's ratio 0.3 and mass density 0.001. Create a homogeneous section with this material property and assign it to the part.

3. Create an instance of the part in the Assembly module

4. In the Step module create a general static step with time period 1s. Make sure NLGEOM is off (a geometrically nonlinear calculation will make the optimizer run very slowly).



5. In the Load module, fix both the horizontal and vertical displacements for the small segment of the boundary on the bottom left of the rectangle, and apply a traction boundary condition with 100MPa magnitude vertically downwards on the segment at the top right of the rectangle.

6. In the mesh module seed the part with a 0.1mm mesh size and mesh it (use the default element type).

7. The topology optimizer is set up in the Optimization module, as follows:

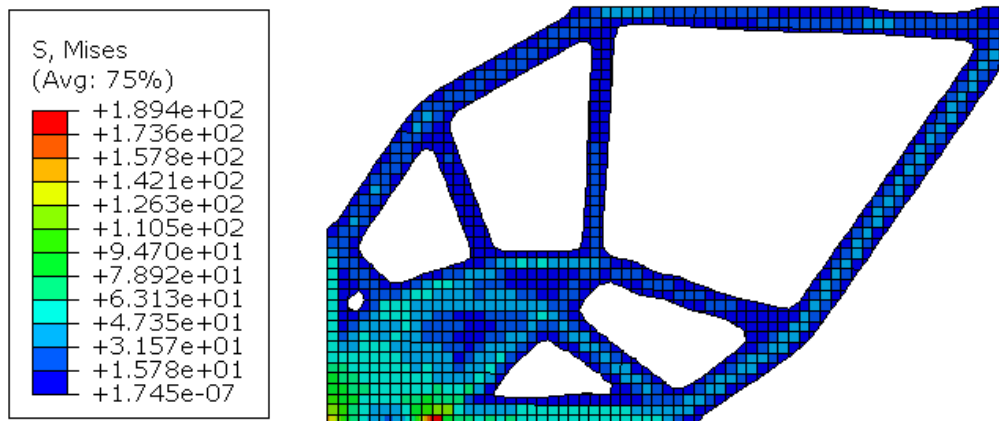
- Use Task>Create to create a new optimization task. Select Topology optimization, then in the Edit Optimization Task menu check the options to freeze both the load regions and the boundary condition regions (this will ensure that the points where the solid is loaded and constrained do not get moved during the optimization).
- Use Design Response > Create and select Single Term to set up quantities that will be calculated and used to specify the optimization problem. Select Whole Model from the buttons below the viewport (this will attempt to change the geometry of everything). Then select Volume from the menu. Repeat this process and select Strain energy to create a second design response that will compute the strain energy of the block.
- Now define the objective function for the optimization: Objective Function> Create, then in the menu select Minimize, and click the empty box below Name and select D-Response-2 from the drop-down menu. This will set up the optimizer to minimize the strain energy (which will maximize the stiffness of the part. If you run a problem with displacement boundary conditions instead of force boundary conditions you should maximize the strain energy to maximize the stiffness.) Press OK.
- Now we will set up Tosca to reduce the volume of the part. Select Constraint>create, press Continue, then in the Optimization Constraint menu select D-Response-1 from the dropdown menu below Design Response (this will enter a constraint on the volume); check the radio button next to 'A fraction of the initial value <=, and enter 0.25 in the box next to this. This will cause Tosca to search for the stiffest possible part with 25% of the volume of the rectangle.

(you may wonder why we chose to make the stiffness, and not the volume, the objective function. It is possible to do the optimization that way as well, but it's hard to know what to do with the stiffness. If you know what target stiffness you need, you can enter it as a constraint. But here, we don't know

8. The optimization is run in the Job module.

- Use Optimization > Create, and reduce the number of design iterations to 30. If you have a computer with multiple processors you can use the Parallelization tab to run the job on multiple processors (optimization, especially in 3D, is slow).
- Now select Optimization > Submit to run the analysis
- Use Optimization > Monitor to track the progress of the computations.
- Once the analysis is completed, use Optimization > Combine > Opt-Process-1 to combine all the separate odb's from each different design iteration into a single one (for visualization).

9. In the Visualization module, use File > Open ... You should see a directory called Opt-Process-1 in the ABAQUS working directory. Inside this directory there should be a subdirectory called TOSCA_POST. Open this directory and open the odb file. You can use the arrows to step through the design iterations. The last frame can display stress and displacement contours (they are not saved for the intervening steps by default, but you can set up ABAQUS to save data for each iteration when you create the 'optimization process' in the job module.



10. You often want to save the part geometry from an optimization run for further editing (you will want to smooth out the boundaries) or further analysis. You can do so by going back to the Job module, then select Optimization > Extract. You can save a .inp file, or a .stl file for your part. For a 2D part, the stl file can be imported and used to create a new geometry easily (this is possible with 3D parts too, but it's much harder to convert them to a geometry that can be meshed). See the ABAQUS CAD import tutorial for more information about how to do this.