Tutorial-

Tetrahedral-meshing of a Steering Wheel

(Written by Moritz Günther, Student at Altair)

Updated to 2017.2

by

Premanand Suryavanshi

1. Click on *file → open → steering_wheel_tetra.hm.* Now your model is in the window.
2. Under the window is the Geometry Color Mode. Choose *By 3D Topo* like in the picture below.


   The green color indicates that the CAD model is a solid.

3. Before we start to mesh the steering wheel, we are going to trim a circle at its center (the circular surface will be used later in order to apply "dummy loads".

A. Go to *Geometry → Edit → Surfaces → Trim with Lines.*
Use the column in the middle and select the two surfaces like in the image on the right first. Then pick the circle as trimming line. Now you have to define a vector or you simply decide for the z-axis.
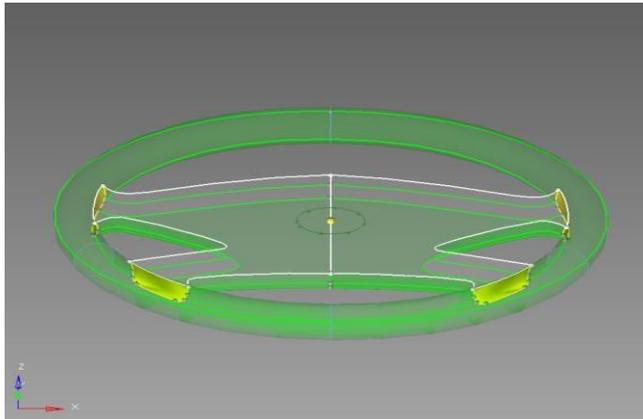


After trimming the entity should look like this:
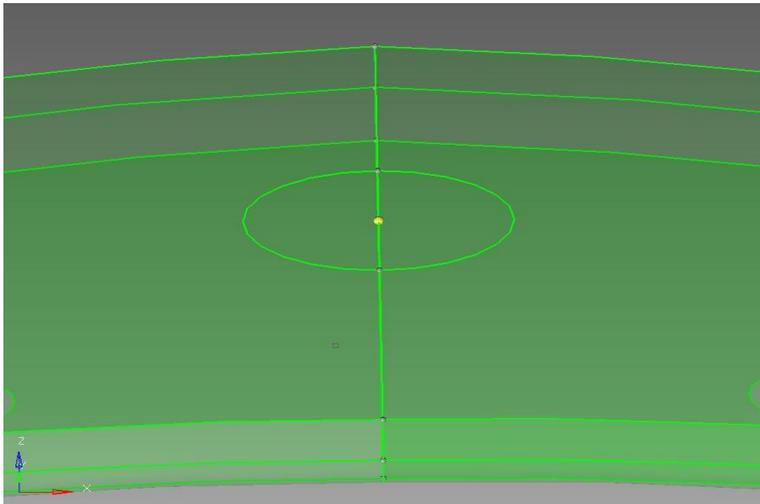
The nodes of the tetra-elements would now be enforced to follow the trimmed green edge (no control on the orientation of the elements off the surface).

B. Alternatively, trim the entire solid with respect to the circle using geom > solid edit panel > trim with lines, using sweep solid option. First pick stroke for solid selector, then pick circle for line selector and choose vector as Z- axis, and click trim. This will then create a new solid (bounded by yellow surfaces) as shown below. On which the nodes of the tetra elements will be placed.
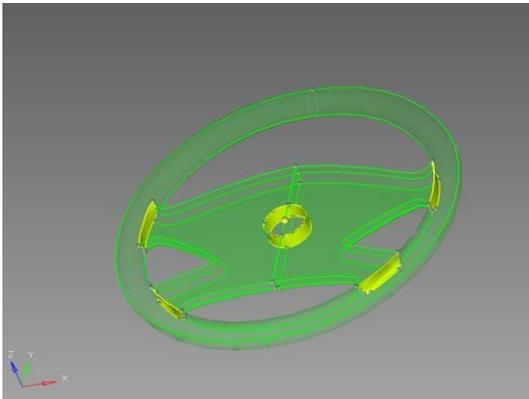

4. Trim the entire model into half, since it is symmetric. Use solid edit panel > Trim with Surf/Plane. Mask half of the model to make meshing easy.
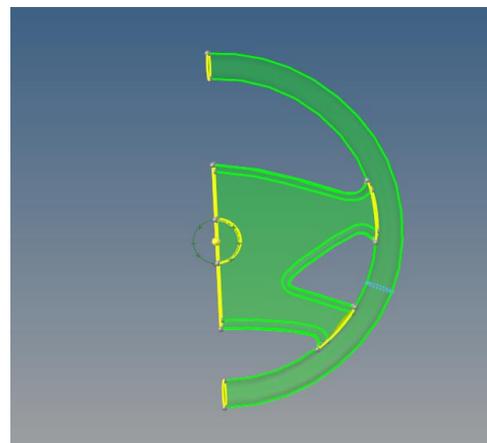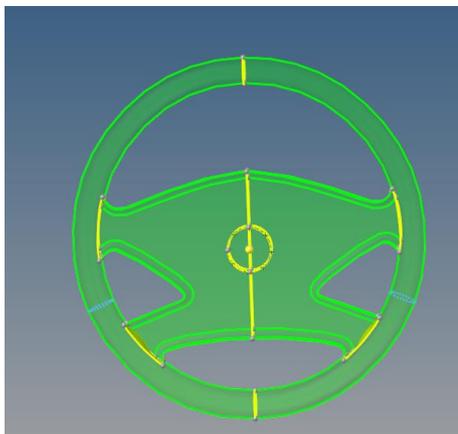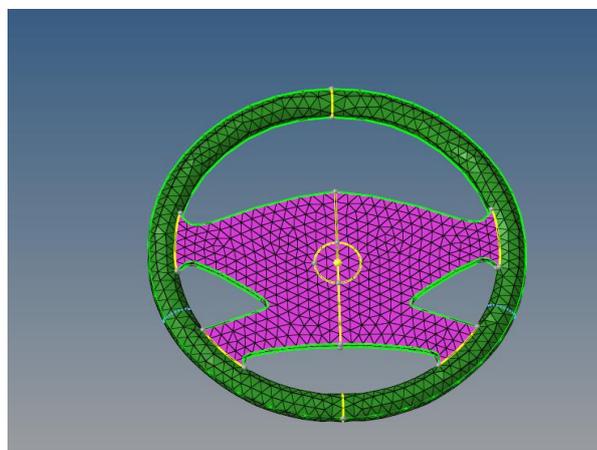
5. Go to *Mesh → create → Tetramesh* and choose the menu *Volume tetra*. Choose the component named *wheel* and specify element size 1. Click on *mesh*.
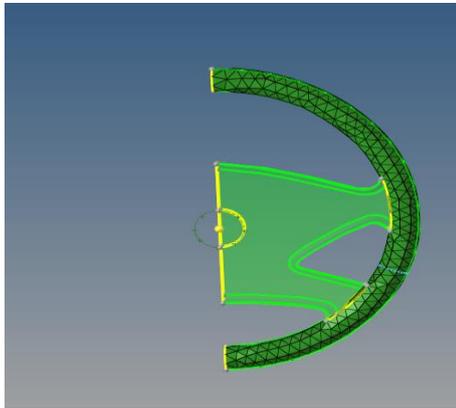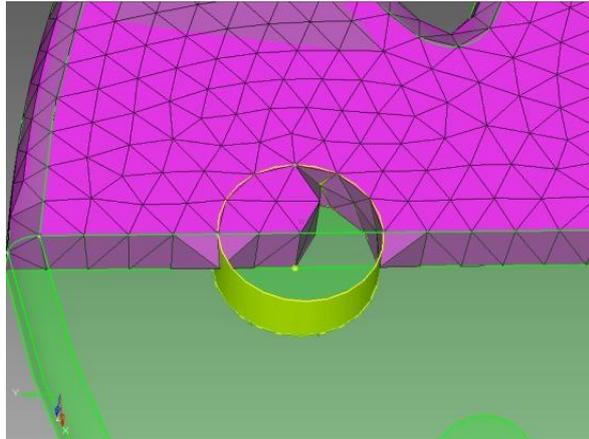

Then go to the Model Browser and make the component named *stroke* current. Mesh the stroke with element size of 1 mm.
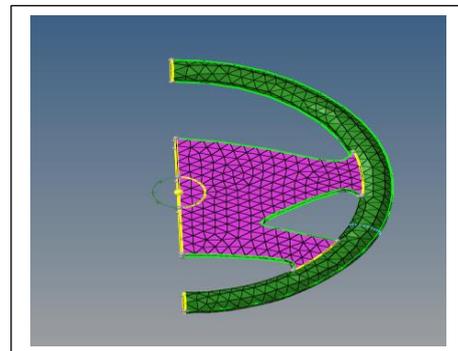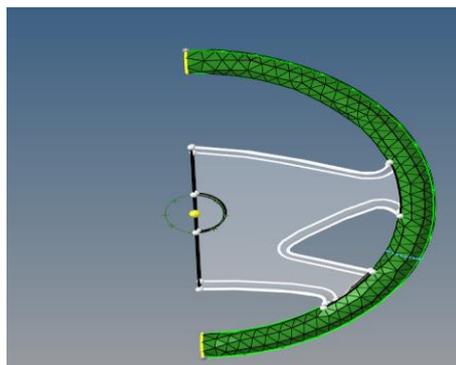
6. Now reflect the mesh elements to other half of the model as shown.

After masking some elements from the middle of the circle, you can see how the elements follow the trimmed plane/surface.

Setting up a linear static analysis (Note, loads and constraints are conceptual)

7. Make sure that the User Profile is set to *optistruct*.
8. Right click in the Model Browser and select *Create → Material.* Type in *steel* and choose *MAT1*.

   Repeat step 6 and create a material collector named plastic.

9. Right click in the Model Browser and select *Create → Property*. Type in the name *wheel* and select the category *Material*. Then choose *plastic* and click Create

   Repeat step 7 in order to "connect" the property collector named *stroke* with the material *steel*.

10. Right click in the Model Browser on *Component → wheel → Assign*, as property collector choose *wheel*.

    Repeat step 8 for the *component* named *stroke* → select *Assign* and choose *stroke.*

11. Now it is time to define the material properties: In the Model Browser go to *Material → steel* and click with the right mouse button on it. Select card edit. Select *E* (Young's modulus) and type in 2.1e+05 N/mm^2, click on *NU* (Poission ratio) and type 0.3.

Repeat step 9 in order to define the material properties for *plastic:* *E* 3000 N/mm^2 and *NU* 0.38.

10. Next we define (conceptual) forces and constraints. Go to the Model Browser, right click and choose *Create → Load Collector*. Type in *forces* and click *create*.

Repeat step 10 and create a load collector named *constraints*.

11. In the Model Browser right click on *forces* and choose *Make Current*.

Go to *Analysis → Forces,* choose the same nodes like in the picture below, use the *z-axis* and type in 50 N as *magnitude*. Click on *create*.
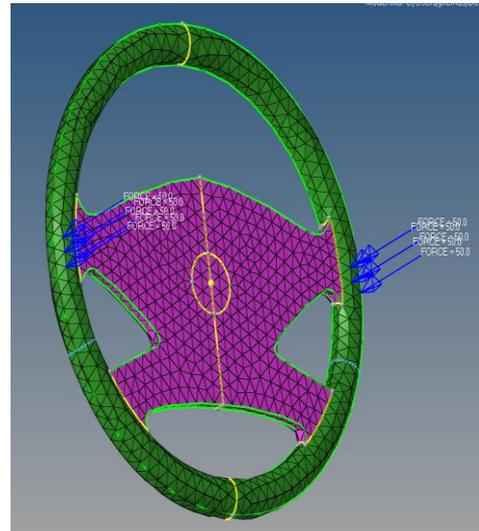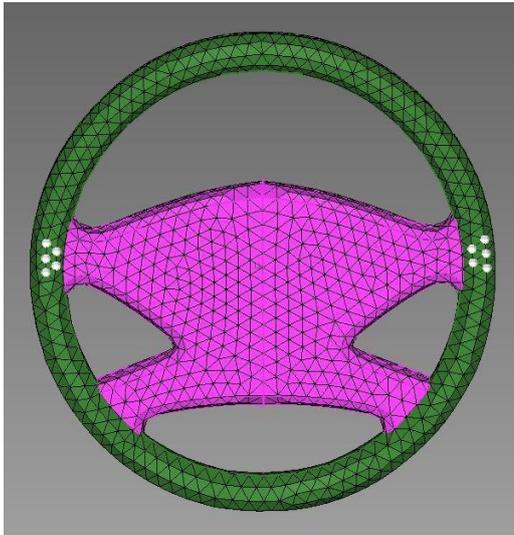
12. Make the load collector named "SPC" current and go to *Analysis →*
    Constraints. Choose the nodes which are on and inside the trimming circle
    and click on create.
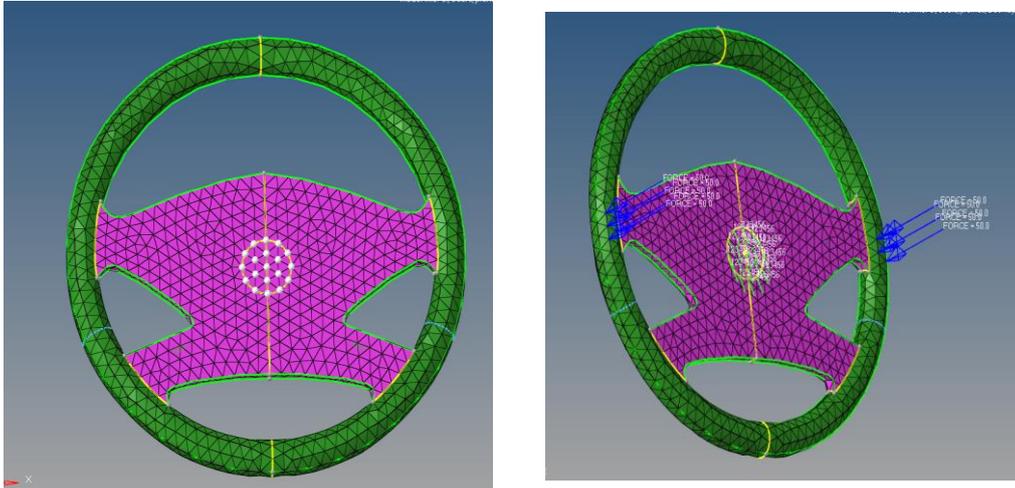
13. Go to *Analysis → Load steps* and type in *steering_wheel* (name of the
    load step). Make a check at *SPC*, click on the empty field next to it and
    select the load collector *constraints*.
    Make a second check at *Load* and select *Forces*.

    Now your model would be ready for a static analysis.