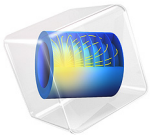


Virtual Operations on a Wheel Rim Geometry

Introduction

CAD geometries, especially those imported from CAD design software, may contain small features, such as faces or edges, that could significantly increase the number of mesh elements in the model. An efficient modeling approach is to avoid such small features by using virtual geometry operations to "hide" them from the mesher. A benefit of using virtual geometry operations is also that the curvature of the geometry is preserved, which could be important for certain physics applications, such as stress analysis.

In this tutorial you prepare a wheel rim geometry for stress analysis by defeaturing it using virtual geometry operations. The stress analysis is described in the model *Submodel of a Wheel Rim* found in the Application Libraries for the Structural Mechanics Module.

Model Definition

The geometry to be meshed is of a ten-spoke rim design, shown in Figure 1.

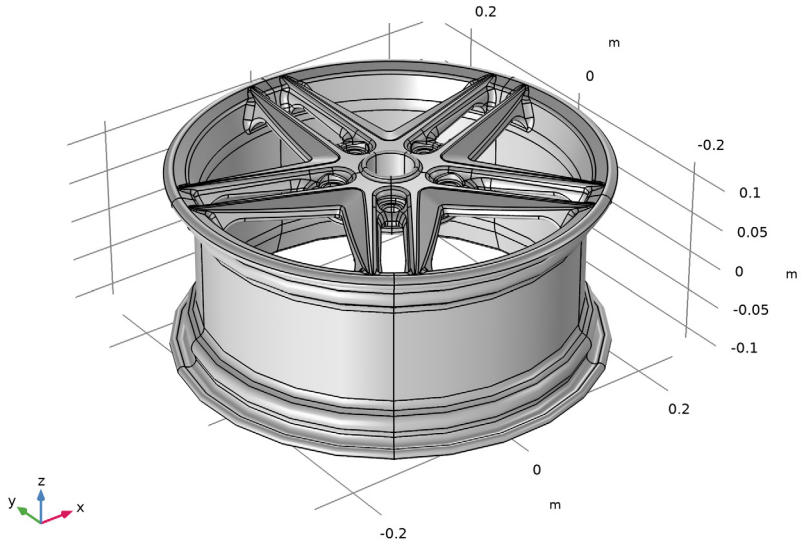


Figure 1: Wheel rim geometry.

To provide accurate results for the stiffness of the rim create a mesh for the analysis using the default normal predefined mesh settings. Apply virtual geometry operations to reduce the number of elements in the mesh.

The mesh generated with the original geometry consists of about 260,000 tetrahedral elements, see Figure 2 below.

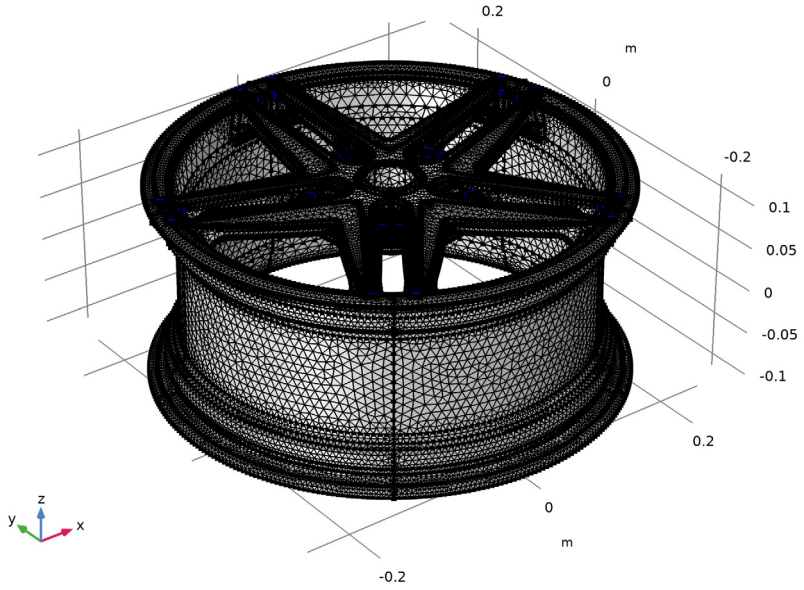


Figure 2: Mesh of the geometry before applying virtual geometry operations

Due to small faces and edges, the mesh generated with a default normal size setting contains warning messages. These messages indicate that some small features could not be resolved by the mesher using the current settings. The warnings can help in locating these geometric entities.

The final mesh, after the virtual geometry operations, consists of about 180,000 tetrahedral elements, see Figure 3 below, which means a reduction of about 30% using the same mesh settings.

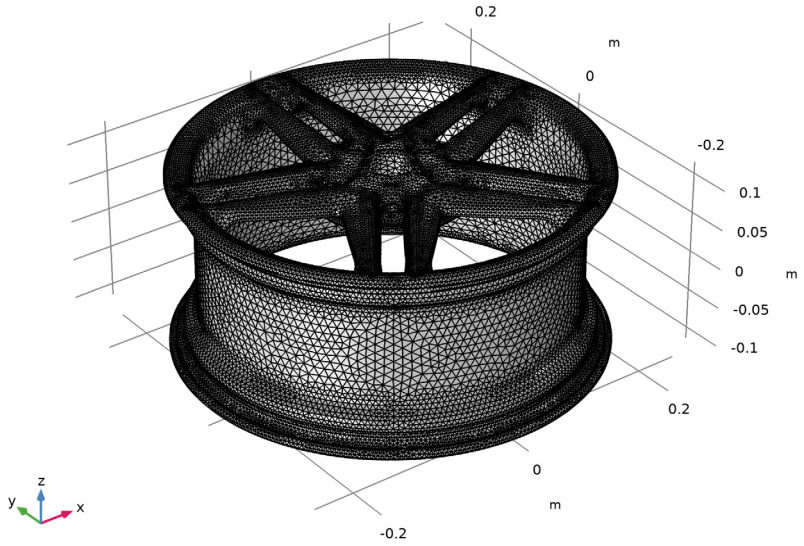


Figure 3: Mesh of the geometry after applying virtual geometry operations

This is a significant reduction of elements which reduces the memory requirements for computing the solution.

Note: The number of elements vary based on the used geometry kernel. If you are using the CAD kernel you can expect fewer elements than the value given in this document.

Notes About the COMSOL Implementation

Virtual geometry operations appear after the Finalize node in the geometry sequence. Virtual geometry operations do not modify the geometry, rather they hide geometric entities, such as faces, edges or vertices, from the mesher. As a result a virtual geometry is meshed, hence the name of these operations.

As a starting point for where on the geometry to apply virtual operations use the information from the warning messages provided by the mesher. Warnings are generated for those small faces or edges that the mesher cannot resolve with the current settings.

For an efficient workflow you can take advantage of the symmetry of the rim design. Rather than creating selections on the full wheel geometry you can apply the virtual operations on only a fifth of the rim. When done recreate the full rim geometry from the segments. The selections of the virtual operations automatically extends to include the full geometry.

Application Library path: `COMSOL_Multiphysics/Meshing_Tutorials/wheel_rim`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 Click **Done**.

GEOMETRY 1

Import 1 (imp1)

The wheel rim geometry has been saved in the COMSOL MPHBIN-format.

1 In the **Home** toolbar, click **Import**.

2 In the **Settings** window for **Import**, locate the **Import** section.

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `wheel_rim.mphbin`.

5 Click **Import**.

Now, generate the full geometry using a transform operation.

Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the object **imp1** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 Click **Range**.

5 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

6 In the **Start** text field, type `0`.

7 In the **Stop** text field, type `4/5*360`.

8 In the **Number of values** text field, type `5`.

9 Click **Replace**.

10 Right-click **Rotate 1 (rot1)** and choose **Build Selected**.

11 Click the **Zoom Extents** button in the **Graphics** toolbar.

Form Union (fin)

Continue with adjusting the repair tolerance for the Form Union operation. This is sometimes necessary when working with imported CAD design geometries.

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Repair tolerance** list, choose **Relative**.

4 In the **Relative repair tolerance** text field, type `2e-5`.

5 In the **Geometry** toolbar, click **Build All**.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

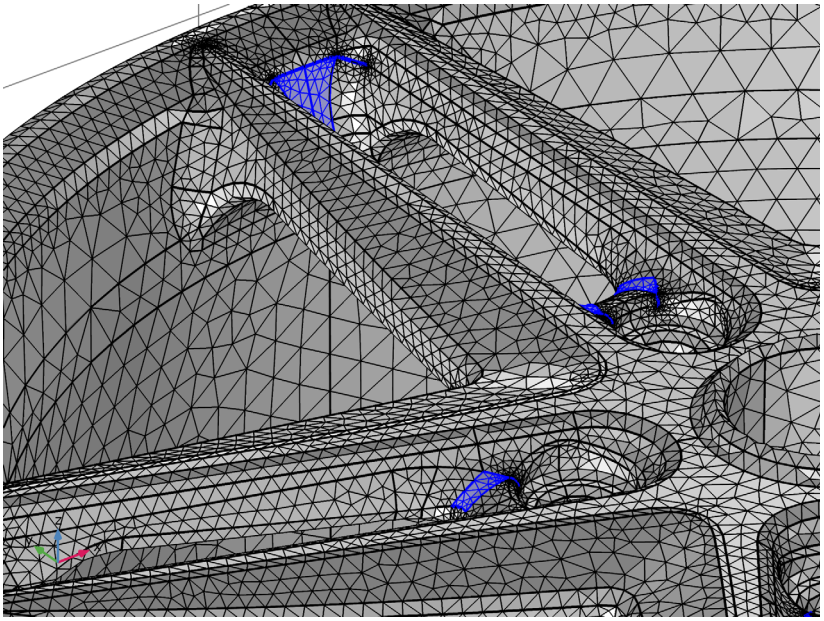
2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Fine**.

4 Click **Build All**.

Warning 1

In the figure below you can see the regions of high mesh density around the bolt holes and in the region of the fillets between the spoke and the rim drum.



These regions indicate small faces or short edges, which you can also locate if you examine the entities reported in the warning nodes for the mesh.

GEOMETRY 1

Using the **Form Composite Faces** operation you will merge the faces of several fillets in the geometry.

First, disable the **Rotate 1** node to work only with the reduced geometry, which will limit the number of selections you have to create for the virtual operations.

Rotate 1 (rot1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Rotate 1 (rot1)** and choose **Disable**.

Form Composite Faces 1 (cmf1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Faces**.

To add faces to the selection, you can pick faces one by one in the **Graphics** window, or use the **Select Box** tool from the Graphics toolbar to select multiple faces at once. A faster

alternative is to enter, or copy and paste the list of face numbers from the instructions into the **Paste Selection** dialog box.

2 Go to the **Form Composite Faces** window, under the **Input** section click the **Paste Selection** button. In the list enter: 2-9,11-17,19-27, 29, 31-46, 51-110, 112-136, 138-160, 162-197, 199-208.

3 In the **Settings** window for **Form Composite Faces**, click **Build Selected**.

Rotate 1 (rot1)

Before meshing you can generate the full geometry of the wheel rim, by enabling the **Rotate 1** node in the geometry sequence.

1 Right-click **Rotate 1 (rot1)** and choose **Enable**.

2 In the **Geometry** toolbar, click **Build All**.

Form Composite Domains 1 (cmd1)

The geometry now contains interior boundaries, which delimit five distinct domains. As the entire rim is made of the same material, you can create a single domain by the **Form Composite Domains** operation.

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Domains**.

2 Click the **Select All** button in the **Graphics** toolbar.

3 In the **Settings** window for **Form Composite Domains**, click **Build Selected**.

MESH 1

Finally mesh the geometry of the rim.

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.