

Bolted Nozzle Flange - Geometry

Creating the Geometry for the Bolted Nozzle Flange

Created using ANSYS 16.2



The full Bolted Nozzle Flange module can be accessed on edx.org [here](#). (Registration is required)

Learning Goals

In this tutorial, you will learn how to create the geometry used for the Bolted Nozzle Flange problem presented in Module 3 of the Cornell's Engineering Simulations MOOC on edx.org.

You can access the MOOC here: [A Hands-on Introduction to Engineering Simulations](#)

Problem Specification

Check Module 3 of the MOOC

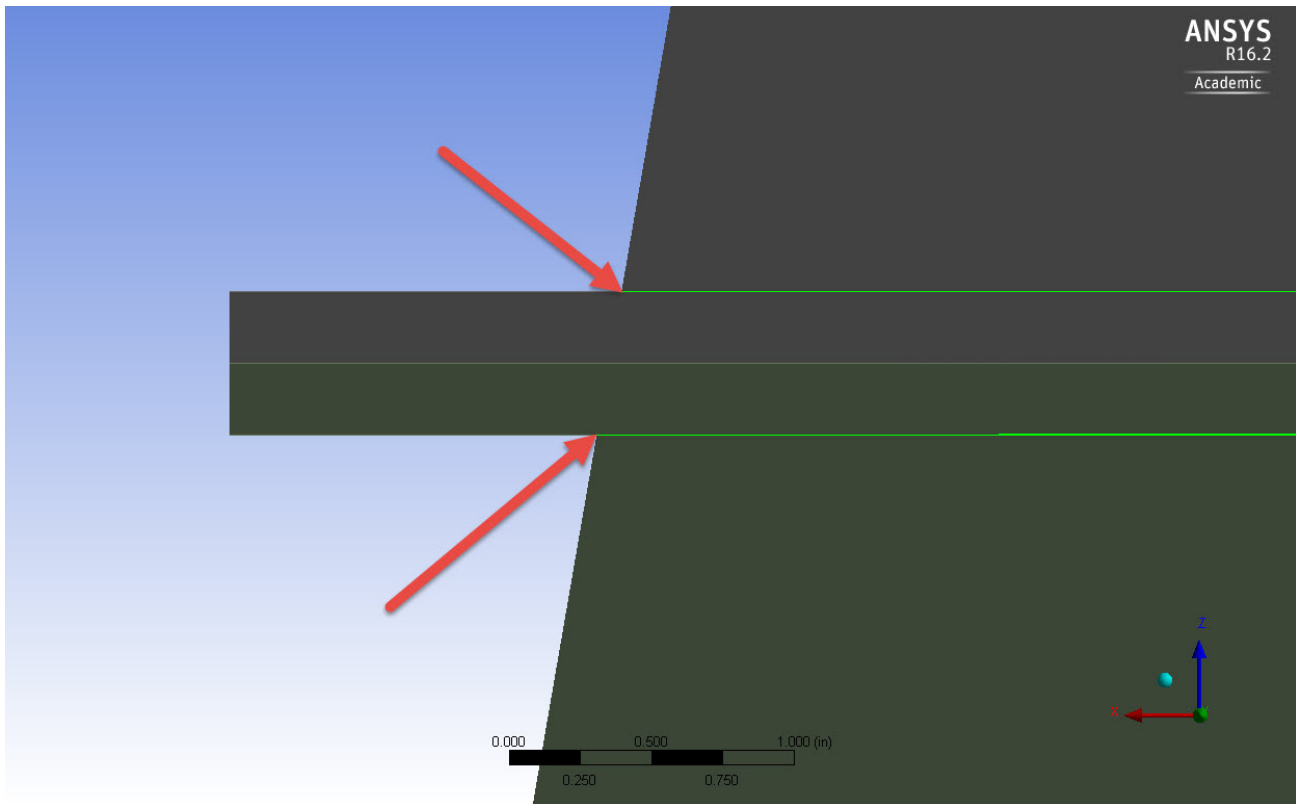
Geometry Creation

The steps on how to create the geometry for the Bolted Nozzle Flange are shown in the following videos.

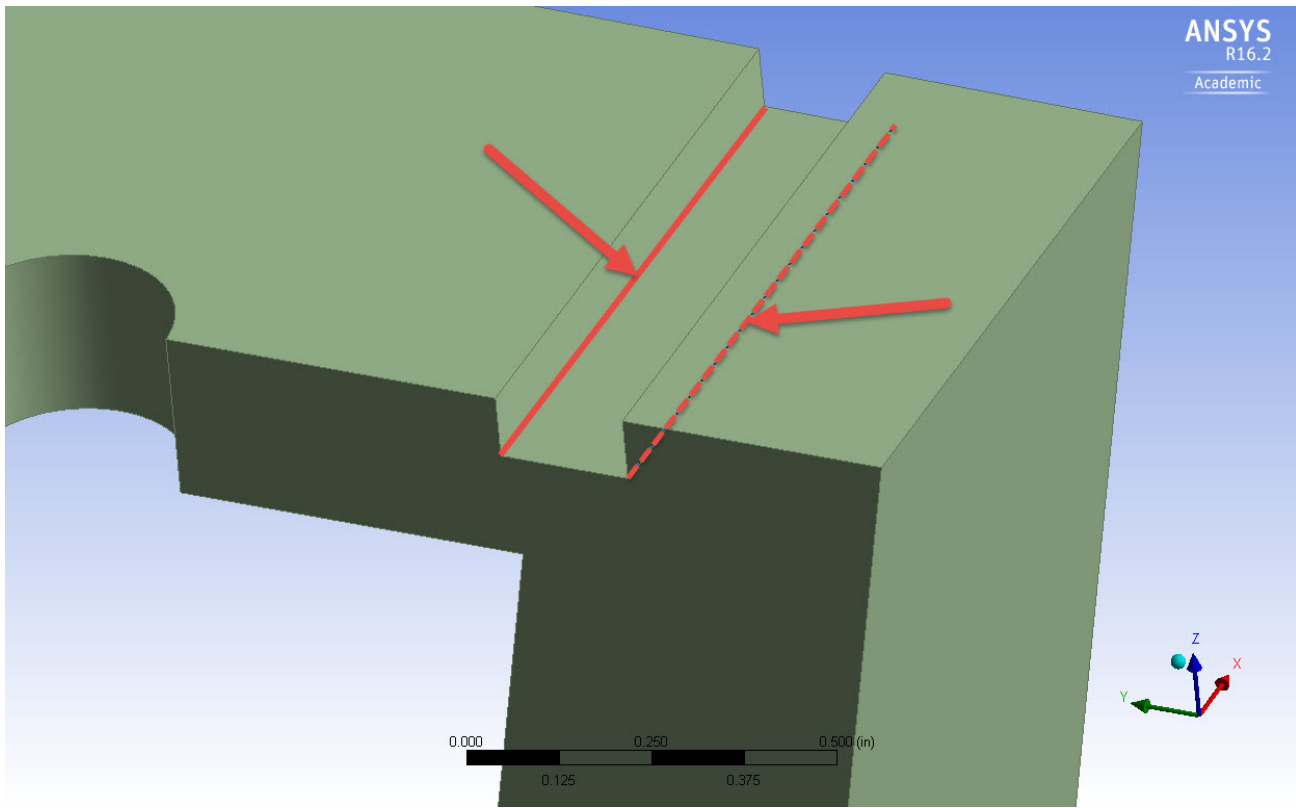
Note: selecting the correct edges

Selecting the correct edges for the Blends can be hard.

For the first blend, you need to select the circular edges around the nozzle; the ones right at the corner between the nozzle itself and the flange. Check the following picture from a side view of the geometry.



For the second blend, you need to select the bottom edges inside the groove. In the picture below you can see them highlighted, but note that to ease the understanding, a "sliced" geometry is shown. Your geometry will look like this by the end of the above video!



Specifying the Contact Regions

[Go to all ANSYS Learning Modules](#)