# Two Gears Simulation

## INTRODUCTION

In order to transfer rotational motion, the teeth of a spinning gear must mesh with the teeth of another gear. In this lab, we will simulate the stresses experienced by meshed gear teeth and compare results to estimates made in practice.

## MODELING THE FIRST GEAR

Save the file as *Double Gear Analysis*. Ensure units are *inches* (**Browser** > **Document Settings** > **Units**).

We will start by creating a gear, copying it, and then meshing the gears.

There are multiple ways to insert gears into Fusion 360: you can model them from scratch, import from McMaster-Carr, or use the Spur Gear Add-In. We will use the Spur Gear Add-In, as it is the most efficient method for this lesson.

*To note, the FEA will typically solve much quicker when using geometry made in Fusion 360 (such as modeling from scratch or using the Spur Gear Add-In), in comparison to imported parts from McMaster-Carr.*

To run the Spur Gear Add-In, select the *TOOLS* menu and click the **Scripts and Add-Ins** icon.
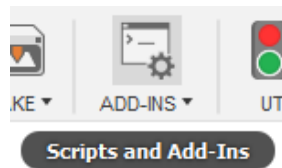


Select the *Add-Ins* tab. Scroll through the list and find the two **SpurGear** scripts. Double-click on either script (they function the same) to create a shortcut.

Select the *SOLID* menu and then click the **CREATE** drop-down. Click the **Spur Gear** Add-In to run it.

Enter the following parameters:

- Standard: **English**
- Pressure Angle: **20 deg**
- Diametral Pitch: **8.0**
- Number of Teeth: **16**
- Backlash: **0.01 in**
- Root Fillet Radius: **0.063 in**
- Gear Thickness: **1.50 in**
- Hold Diameter: **1.0 in**
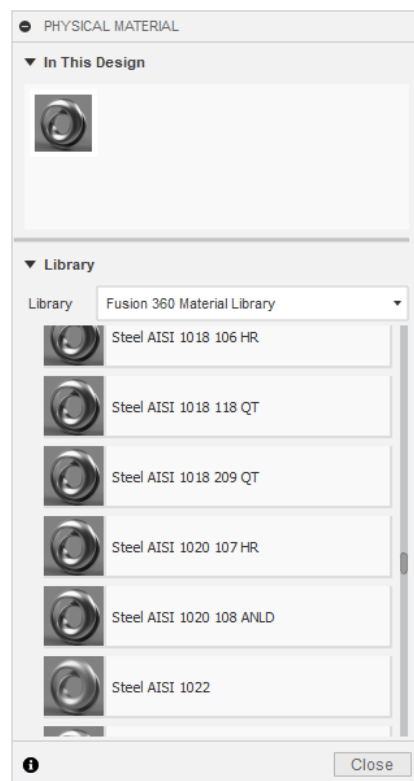
Leave the remaining fields unchanged.

Click **OK** to generate the gear.

To match the specifications, change the material to Steel AISI 1020, either hot-rolled (HR) or annealed (ANLD).

Right-click on the component in the Browser and select *Physical Material.*

Under the *Metal* section, find the appropriate material.



Drag and drop the material onto the gear to change its material properties.

We will now create a copy of the gear and align them so they will contact during FEA.

**CREATING AND POSITIONING THE SECOND GEAR**

From the Browser, right-click on the spur gear component and select **Move/Copy**.

Before moving the gear, check the **Create Copy** checkbox, or this process will have to be redone.
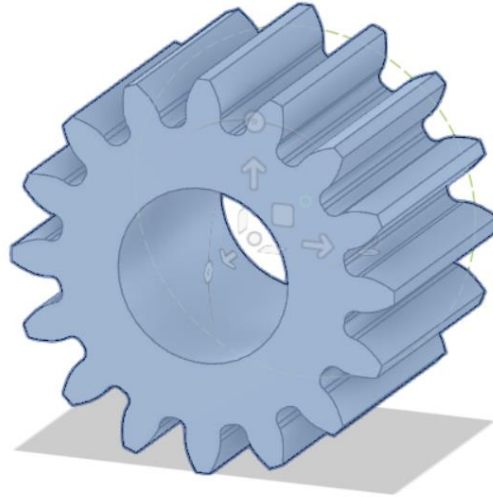


Click and drag the arrow aligned with the x-axis. Move the gear to the *left*, in the negative x-direction, until the dashed pitch circles just kiss eachother.
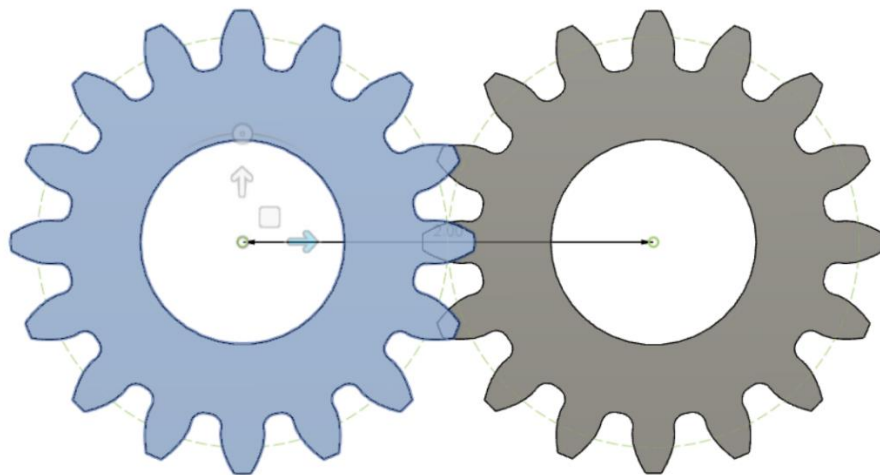


In the dialogue box, the *X Distance* should be −2.00 in.



Finally, change the gear's *Z Angle* to 11.25 deg to mesh the teeth.



*We obtain this angle by determining how many degrees are required to rotate the gear by half a tooth. There are 16 teeth per 360 degrees. Therefore,* $\frac{360\ deg}{16\ teeth} \times \frac{1}{2} tooth = 11.25\ deg.$
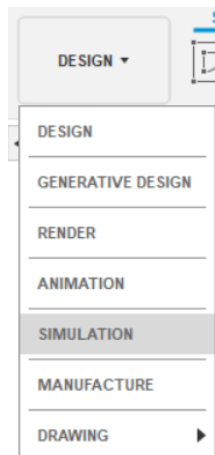
Click **OK** to finish moving the gear.
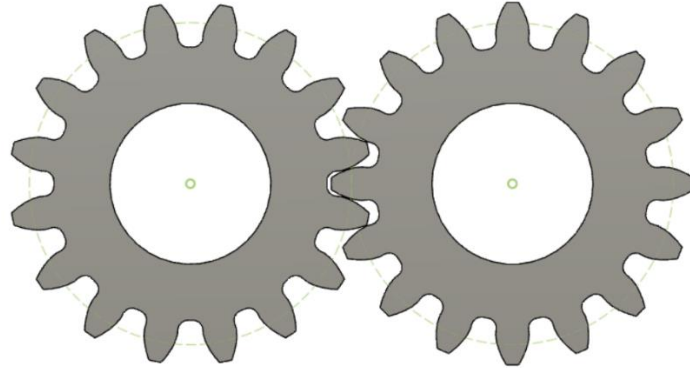


Now that the gear teeth are meshed, we will proceed with the simulation.
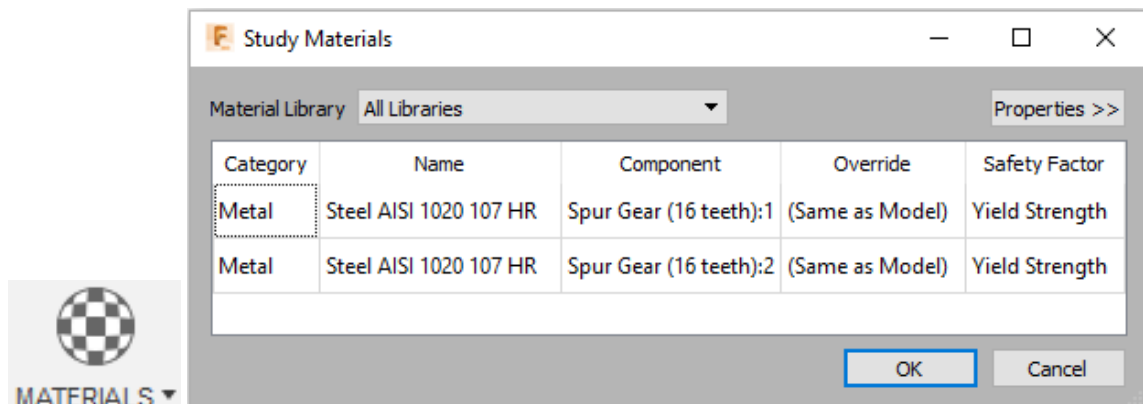
**FEA SIMULATION**

Go to the top-left drop-down and select **DESIGN > SIMULATION**.



Choose **Static Stress** and click **Create Study**.

In the Browser, make sure units are **U.S. (in)**.

Click the **Study Materials** icon and check that the simulation materials are correct.

Click **OK** to confirm the simulation materials.

To simulate the right gear applying a torque on the left gear, we will fully fix the left gear and apply a pin joint on the right gear (so it may spin in the axial direction). Then, we will apply a tangential load on the right gear, which will rotate the gear into tooth contact with the fixed gear on the left.

Click the **Structural Constraints** icon and constrain the left gear hole in all directions.
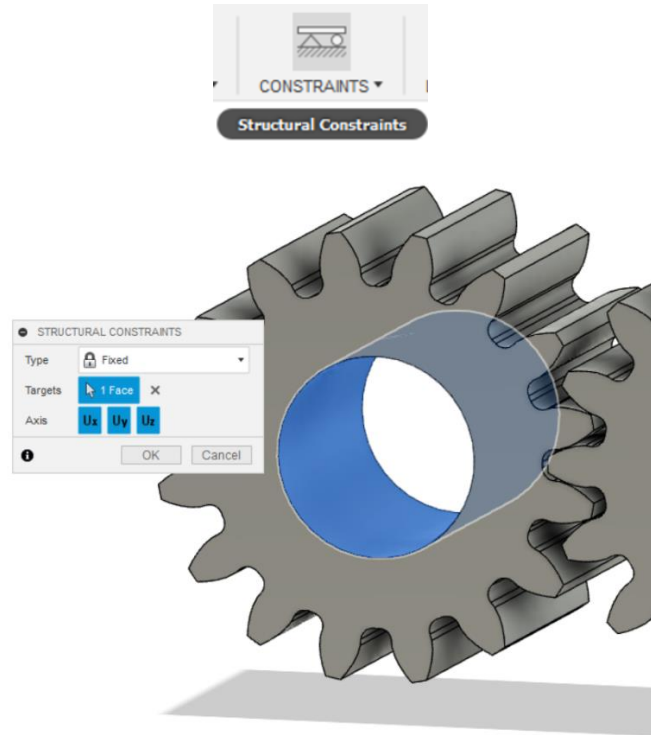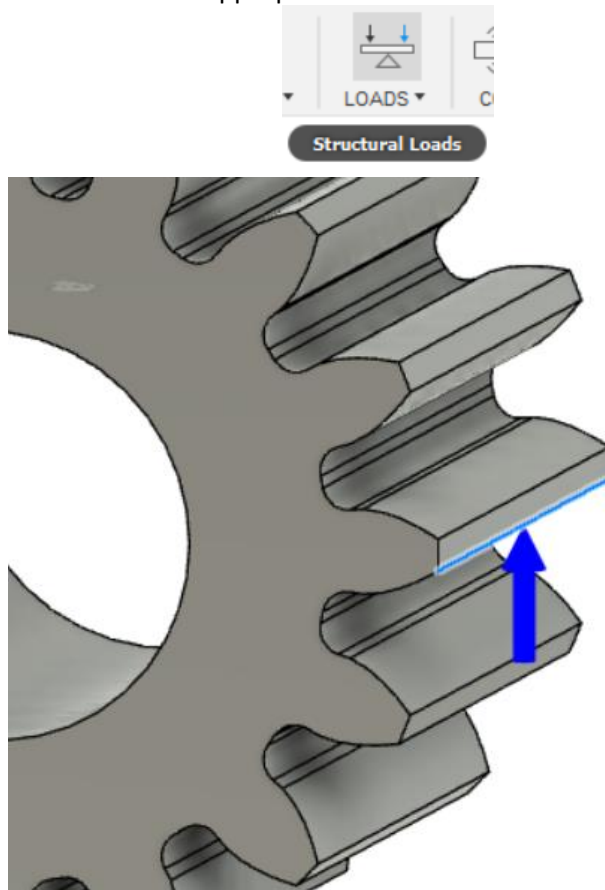


Click **OK** to add the constraint.

Now, apply a *Pin* constraint to the right gear hole. Constrain it in just the *Radial* and *Axial* directions in order to allow rotation.
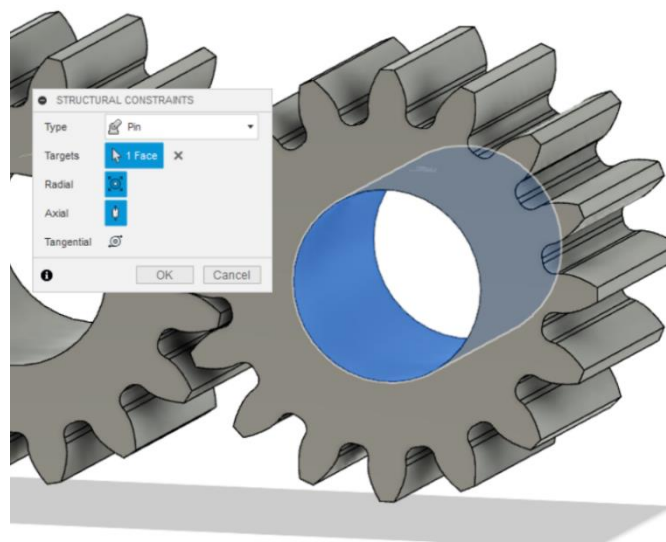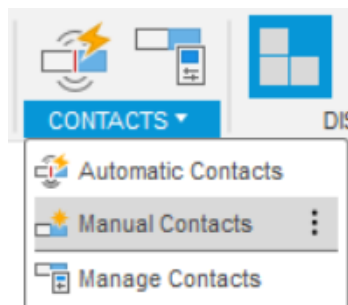
Select the **Structural Loads** icon and apply a *Vector* force on the right-most tooth edge on the right gear. Apply the required load in the appropriate direction.



Click **OK** to apply the load.

We will now add contact sets so the appropriate gear teeth contact each other.

Select **CONTACTS** > **Manual Contacts** from the toolbar. Click **No** when asked to generate automatic contacts.
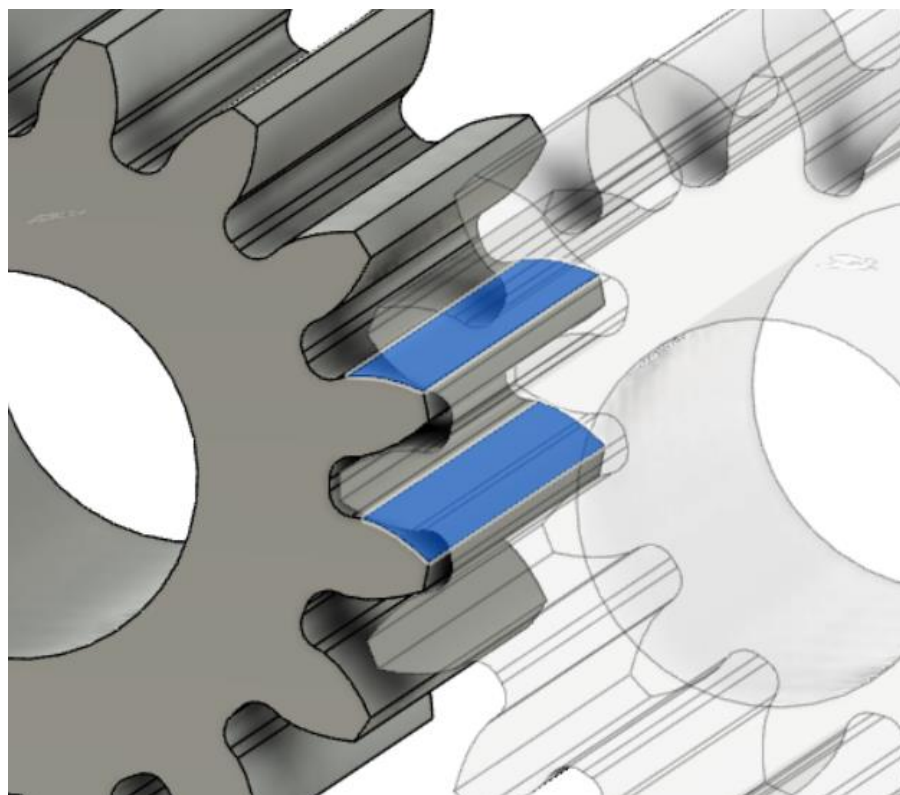


Select the right gear as the *Master Body* and the left gear as the *Slave Body*. As long as *Symmetric Contact* is enabled (it is by default), it doesn't matter which body is the *Master* or *Slave* body.

Select the appropriate locations for *Selection Set 1* and *Selection Set 2* so the required teeth faces will *interact*

**Selection Set 1:**
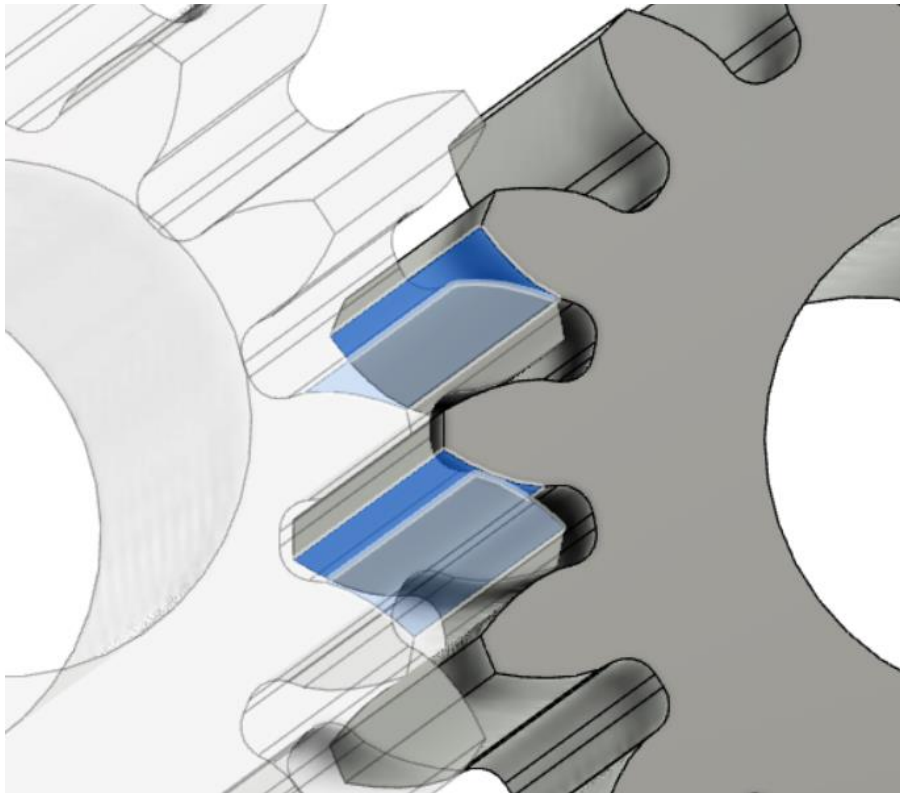


**Selection Set 2:**

Change the *Contact Type* to **Separation** and keep the *Penetration Type* as **Symmetric**.

*Separation contact allows the teeth to initially be at a distance away from each other. When the right gear rotates and the teeth interact, the meshed tooth faces can contact and slide against each other.*

*For more information on different contact types in Fusion 360, click here.*

Click **OK** to create the contact set.

From the Browser, right-click **Mesh** and select **Toggle Mesh Visibility**. A pop-up will appear. Click **Yes** to compute the mesh.
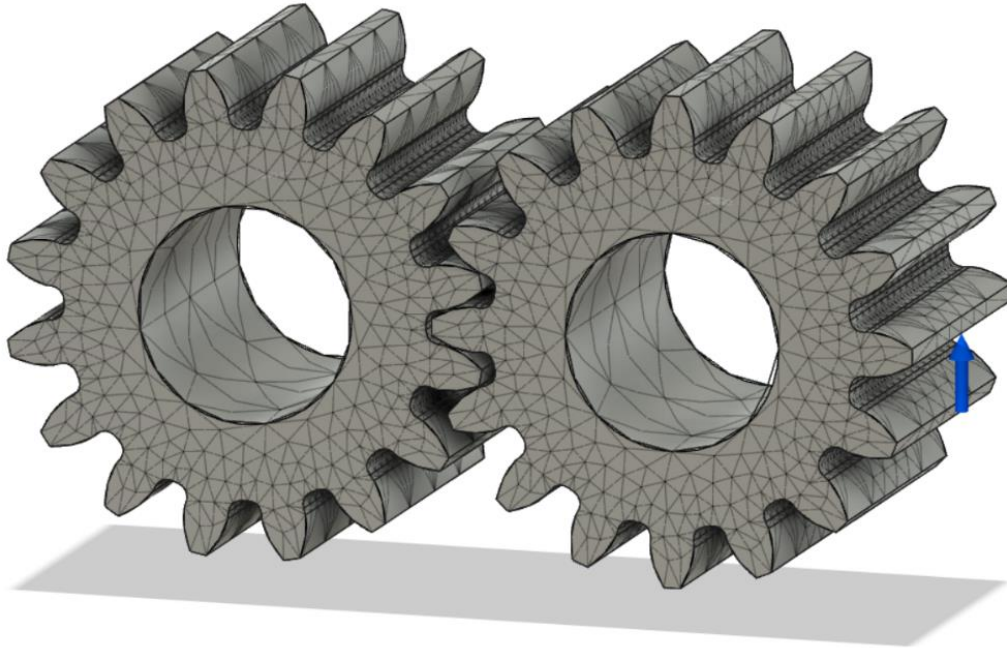


Now, to improve the precision of our results, we will solely refine specific regions of the model. The reason we aren't doing a global refinement is due to the number of elements in the simulation. Decreasing the global mesh would refine regions we have no concern for and greatly increase computation time.

When these gear teeth mesh, we expect to see stress concentration factors in the fillet regions of the contacting gear teeth. To improve results in these regions on the right gear, we will apply a local mesh control.

Select the **MANAGE** > **Local Mesh Control**.
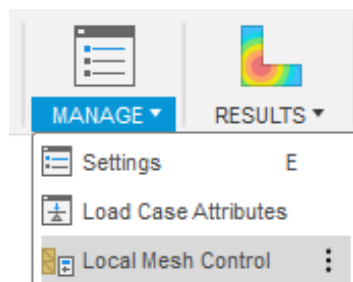
After contacting the left gear teeth, the face regions will experience contact stresses while the fillet regions will experience stress concentration factors. Hide the left spur gear, then click the box next to **Face/Edge Selection** and select the 11 tooth faces shown below.

*Due to their lower stresses, there isn't much need to include the in-between faces.*

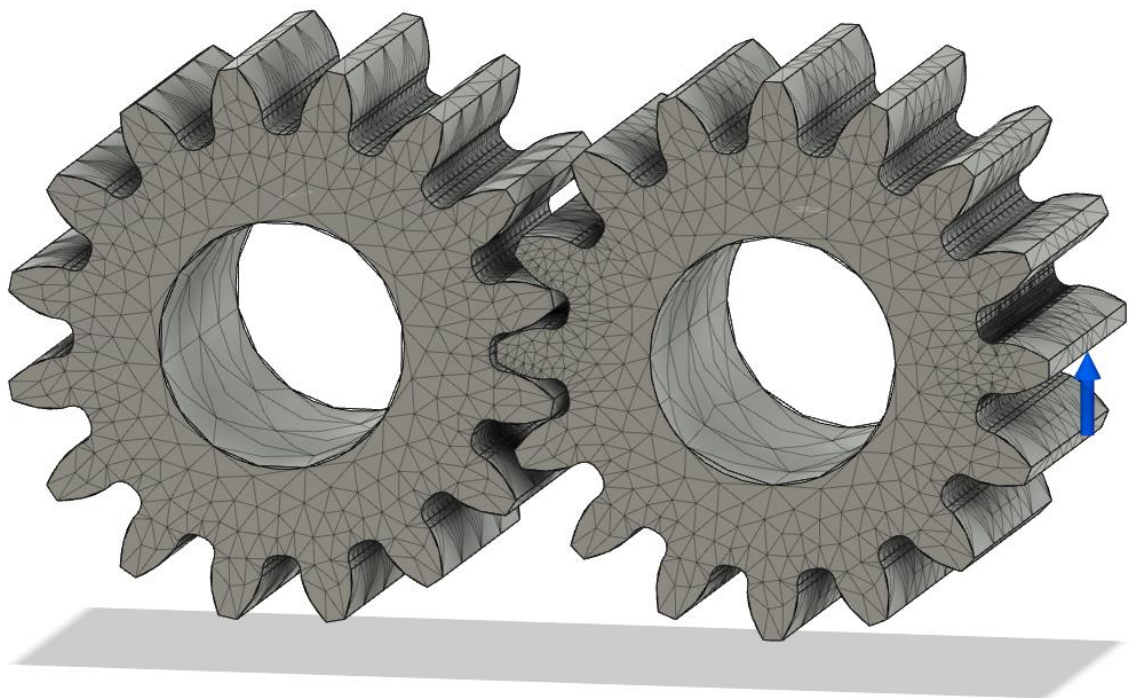| **Top View** | **Bottom View** |
|:---:|:---:|
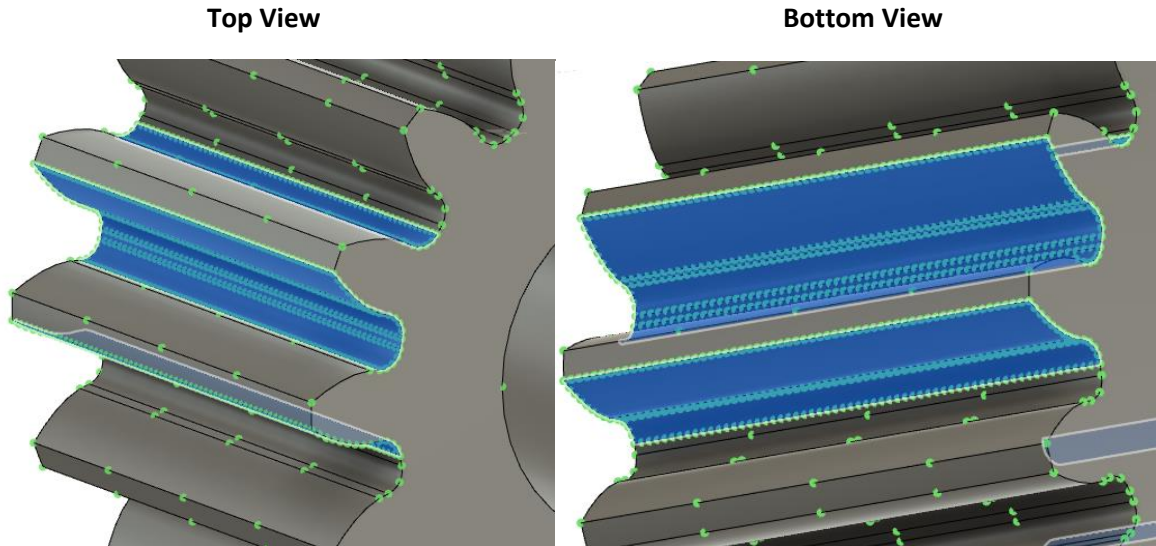


Then, select the high-stress fillet regions on the other side. Including the previous selection, the total number of selected regions will now be 11.
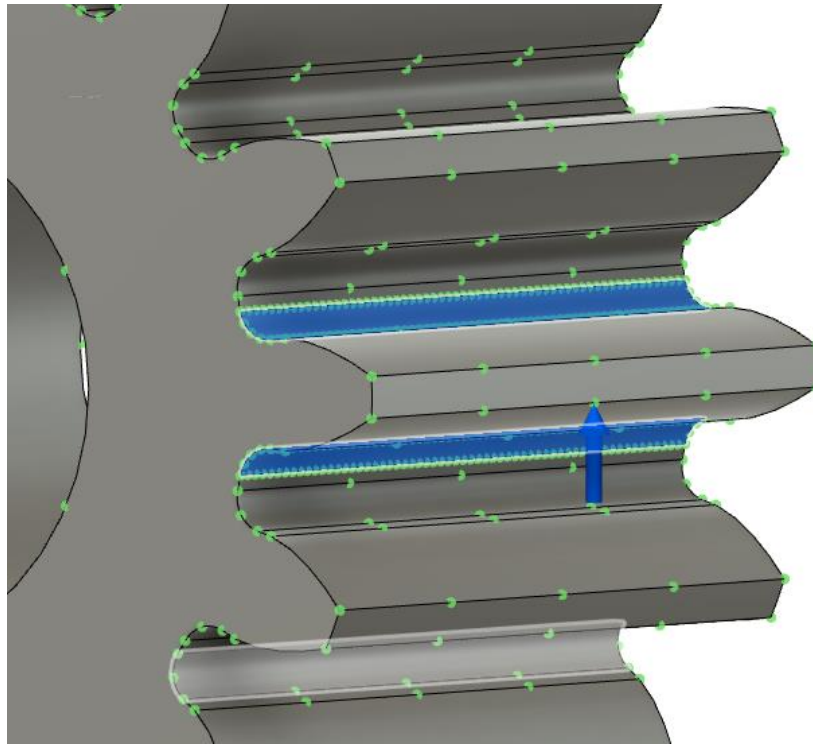


Change the *Length* to **0.025 in**.

Click **OK** to refine the local mesh.

Go to the Browser, right-click *Mesh*, and generate the refined mesh.



It's ok that the *Pre-Check* is yellow  , as the partially-constrained gear will be impeded by the fixed gear.

Due to the vast number of elements in the model, the simulation will take some time (likely approaching 10 minutes).

Before running the simulation, double-check your constraints, applied load, and contact sets.

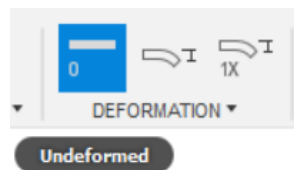After beginning the simulation, click *View Progress* next to the study to see how the calculation process is going.

When the results are complete, close the *RESULTS DETAILS* message.

Click **Safety Factor** and change results to **Stress**, **Von Mises**, **psi**.
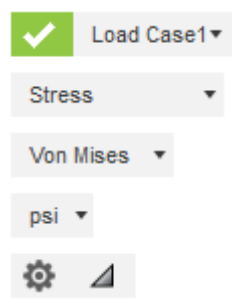


Deformation is exaggerated. Change the deformation scale to *Undeformed*.
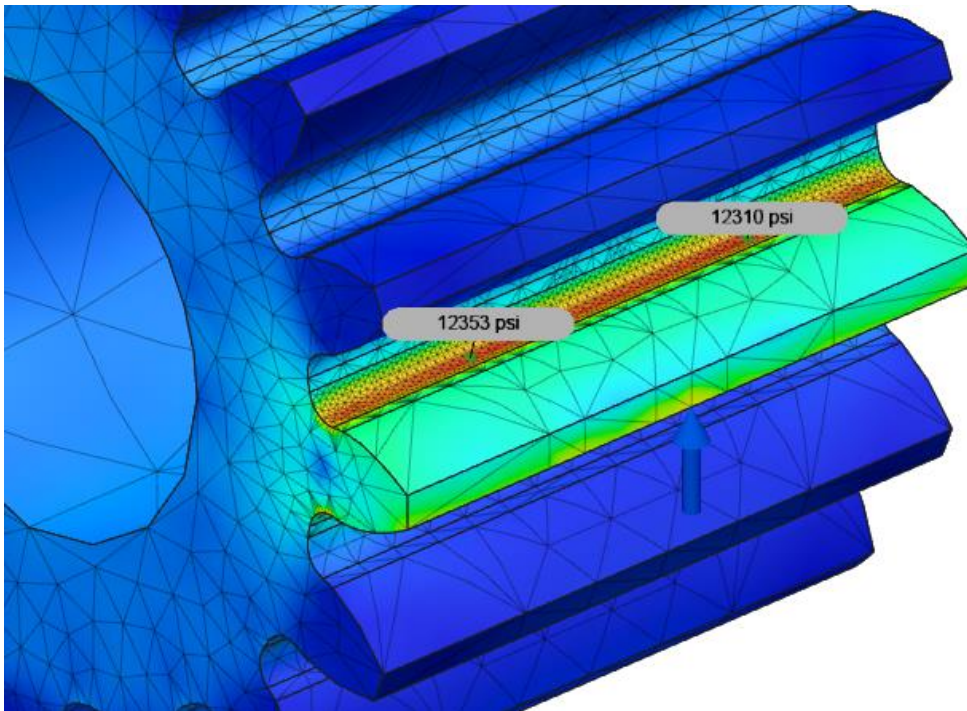


On the right gear, you should be able to see the stress caused by the load on the right-most tooth fillets, along with the stresses caused by the other gear on the left-most teeth.
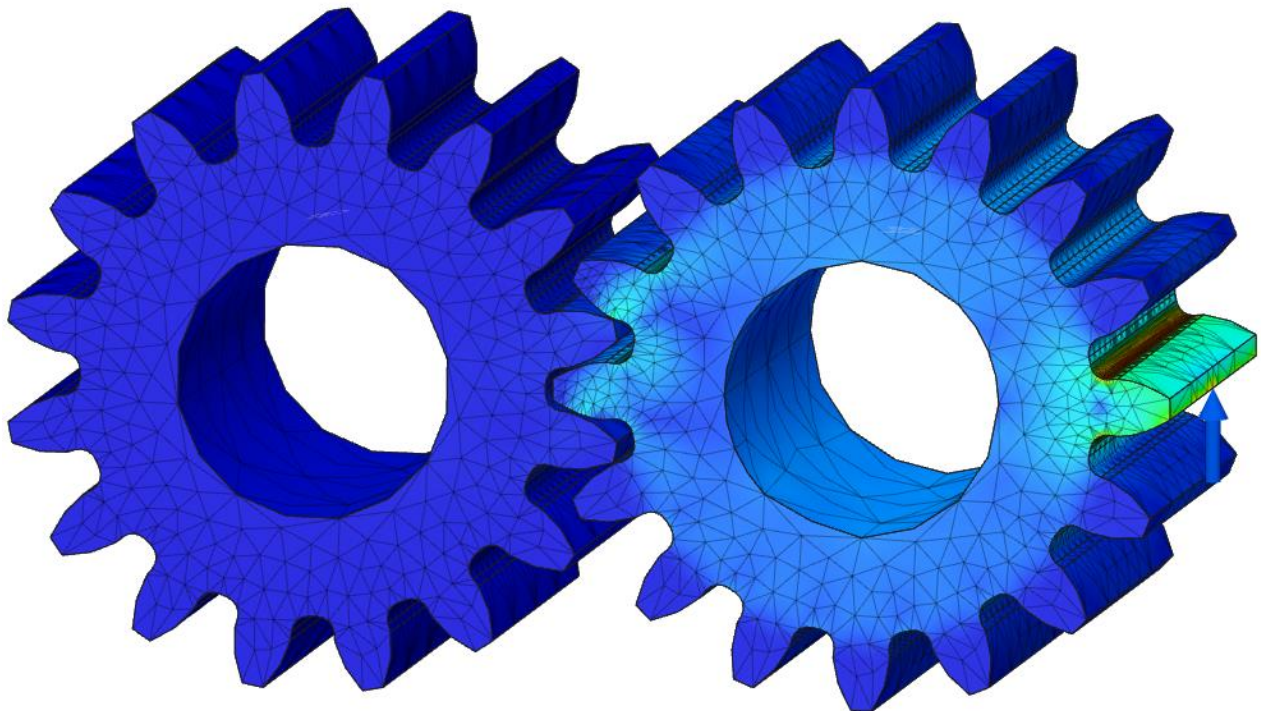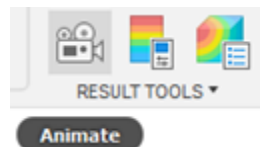
**SUMMARIZING RESULTS**

You may analyze critical regions with a surface probe (*INSPECT* > *Create Surface Probes*).



For this type of study, it would be interesting to see an animation of the simulation. Select the *Animate* icon.



Check the box next to *Two-way* and change the *Speed* to *Fast*. Select *Play* to animate the simulation.

If you'd like to obtain a Results Summary, along with images of your FEA, click the *Report* icon.
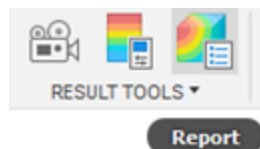


The analysis is complete. Save your work.