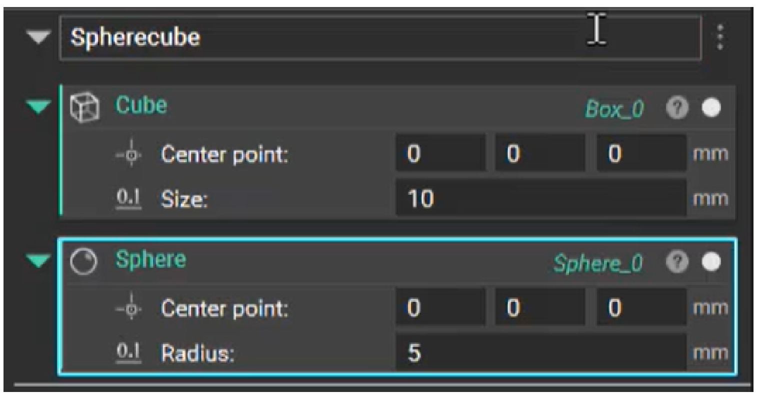
Follow Along: Spherecube and Gear

In this video, we walk through two basic workflows in one Notebook. The first workflow creates a simple Spherecube geometry using nTopology primitives. The second workflow brings in a CAD file of a gear, shells it, then adds escape holes.

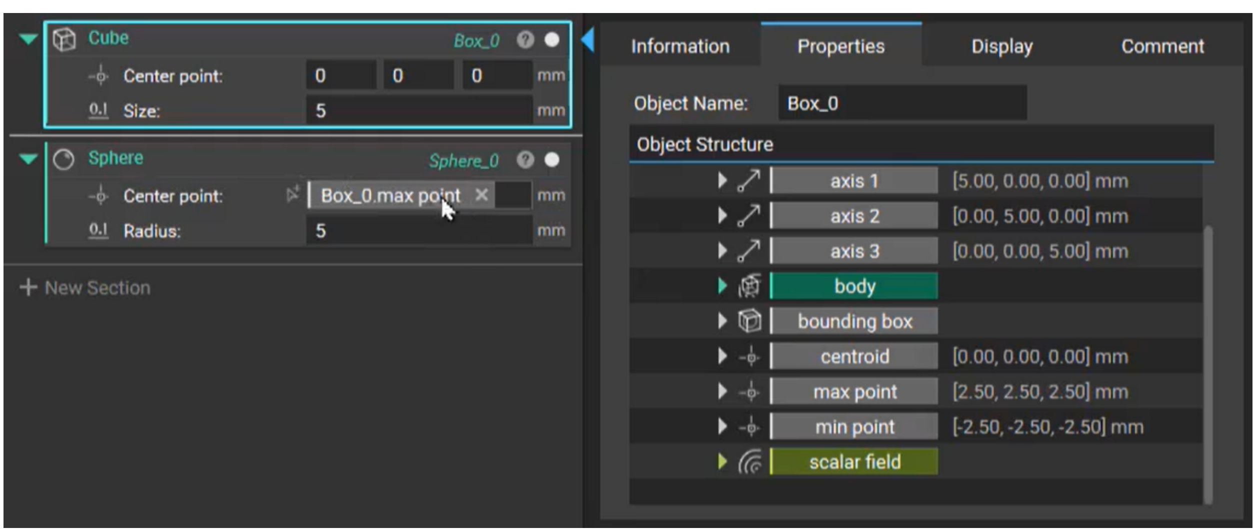
Please download the CAD file below to follow along with the tutorial.

Spherecube

Step 1: Add a **Cube** block and a **Sphere** block to the Notebook. These primitives can be found in the Create Tab. Rename the section Spherecube.



Step 2: Move the **Sphere**'s center point to one corner of the **Cube**. To create a dependency between the **Sphere**'s location and the **Cube**'s corner point, go into the properties of the **Cube** and drag either 'max point' or 'min point' property chip into the Center Point input of the **Sphere**.



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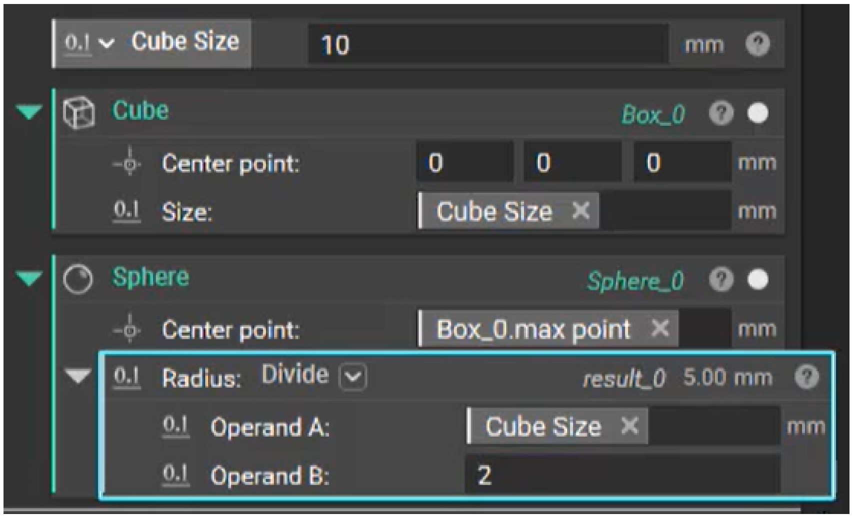
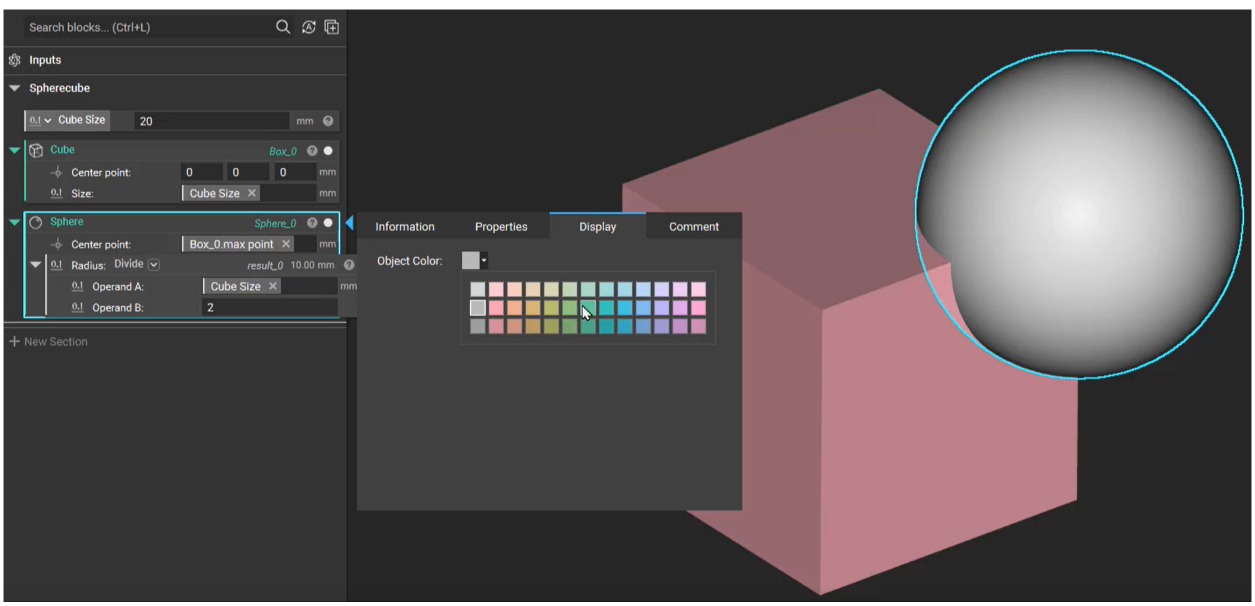
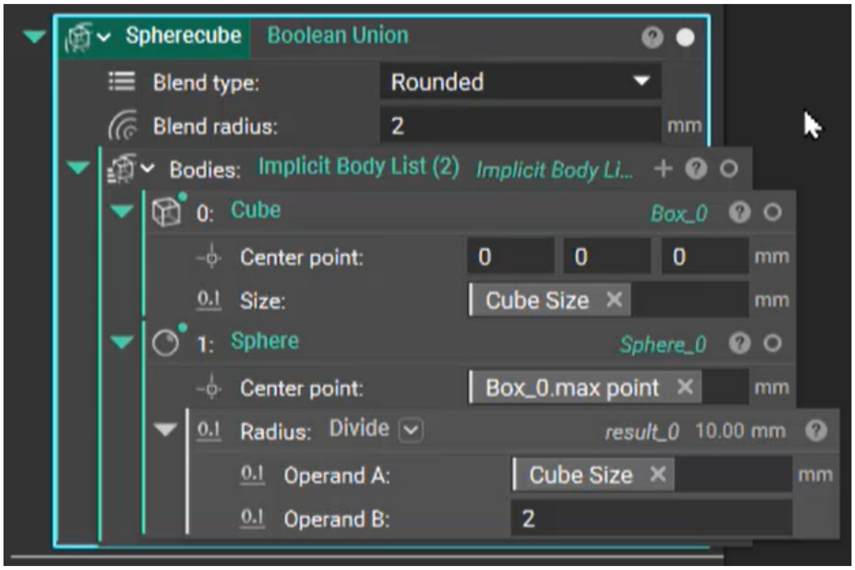
Step 3: Fix the ratio of the size of both objects so that the radius of the **Sphere** is always half the **Cube** size. Use a **Divide** block from the Math Tab as input for Radius. Make the **Cube** Size input a variable called ‘Cube Size’ and use it for Operand A.

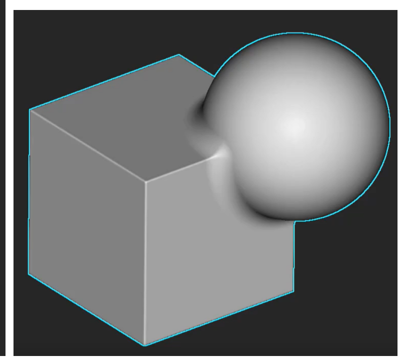


Step 4: Change the display colors of both objects by going to the Info Panel of each block.



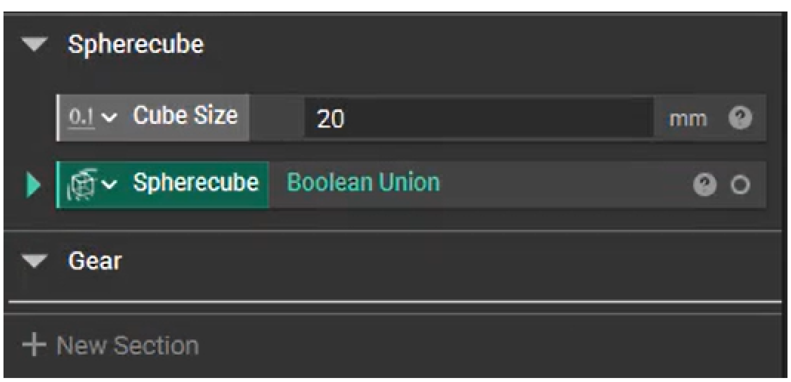
Step 5: Merge the two objects together using the **Boolean Union** block, found in the Modeling Tab. Drag the **Cube** and **Sphere** blocks into the Bodies List, then add a Blend Radius of 2. Make the **Boolean Union** block a variable named Spherecube.





Notice that a new object is created after this step. If you isolate either the **Cube** or the **Sphere** block, the original objects with the changed display colors are still there.

Step 6: Collapse the Spherecube block and turn off its visibility. Add a new section called 'Gear' to start the next workflow.



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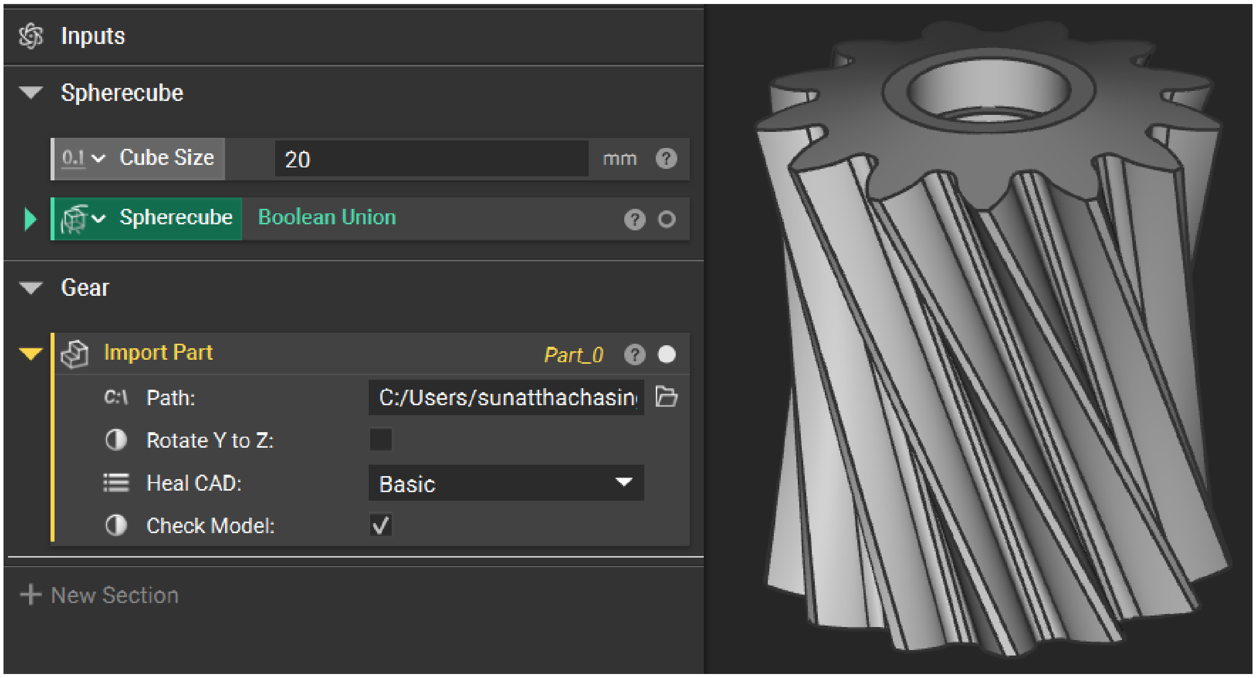
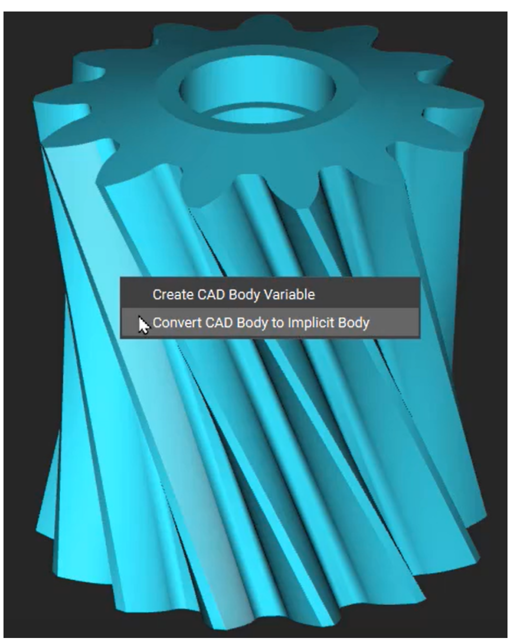
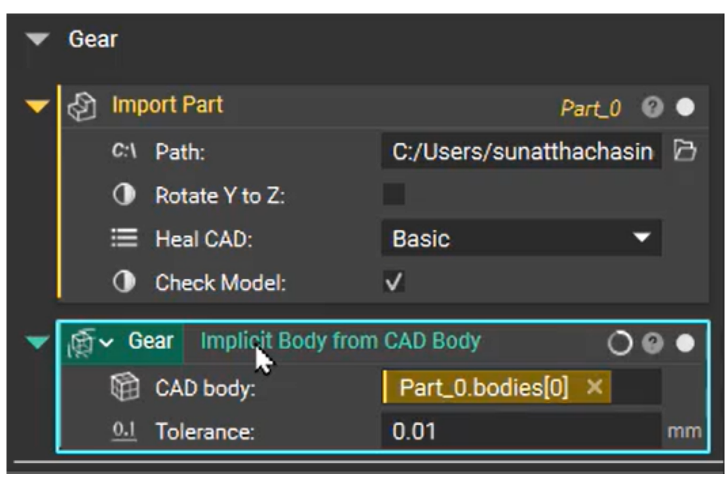
Gear

Step 1: Import the 'Gear' file. If the part is not visible on the screen, press Z to zoom to it.



Step 2: Double-click on the Gear to select the whole body. Right-click and choose 'Convert CAD Body to Implicit Body'. A variable block will appear in the Notebook. Rename this Implicit Body variable to 'Gear'.





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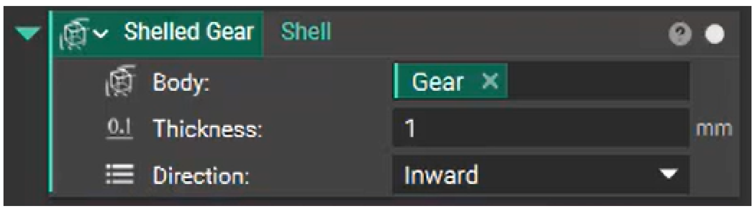
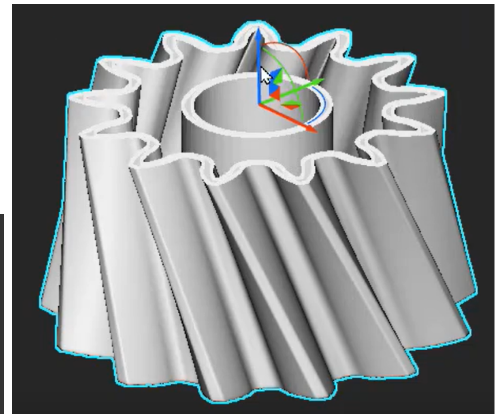
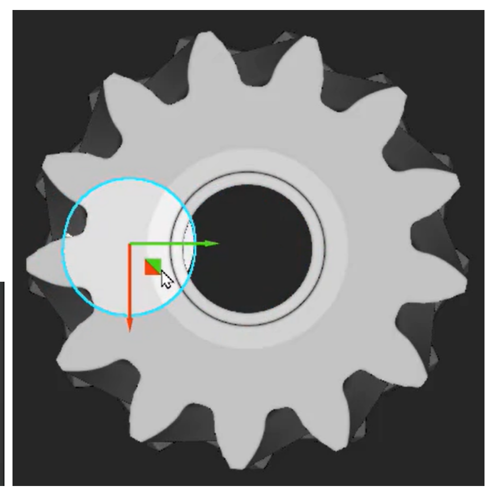
Step 3: Shell the Gear using the **Shell** block found in the Modeling Tab. Drag the Gear implicit body into the Body input and give a thickness of 1 mm in the Inward direction. Make the block into a variable called 'Shelled Gear'. Isolate the **Shell** block and press X to see its section cut.





Step 4: To add holes on the top surface of the shelled gear, create a series of cylinders to be used to boolean subtract from the shell. Start by adding a **Cylinder** block from the Create Tab. Place the cylinder so that it overlaps the part of the shell where material should be removed.

Click on the Point 1 or Point 2 input field to be able to change their location using the axis gimbal. Make use of the viewcube to help adjust the view during placement.



Step 5: Once the two points are positioned at the approximate locations, clean up coordinate values and change the Radius to 2 mm.

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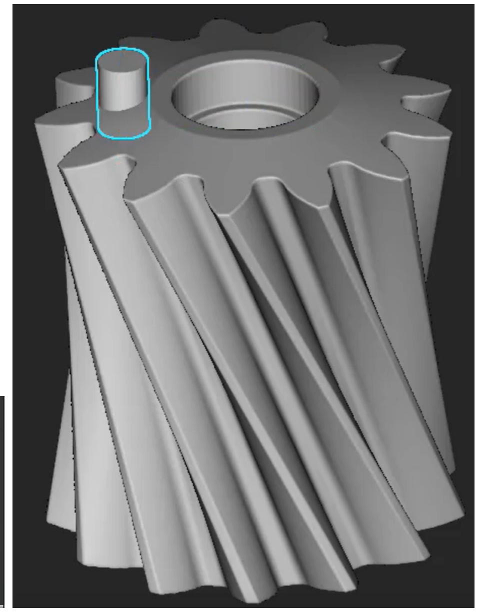
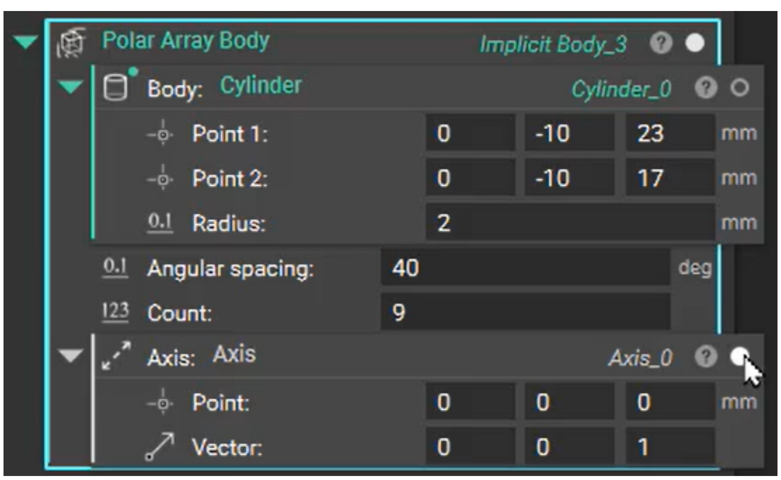
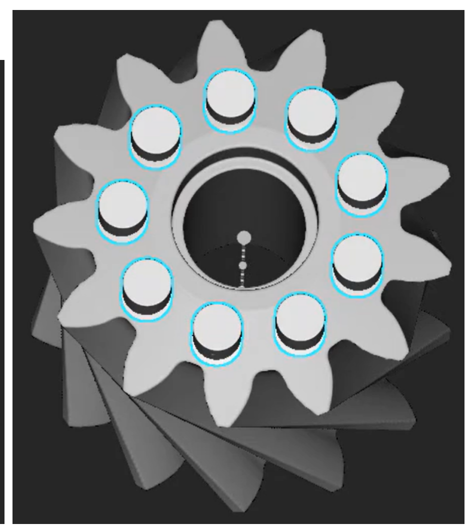


Step 6: Use the **Polar Array Body** block from the Modeling Tab to repeat this cylinder about the center of the gear. Place the **Cylinder** block in the Body input and change the Angular Spacing and Count to 40 degree and 9 respectively. Add the axis of rotation by double-clicking on the Axis input field and selecting **Axis**.

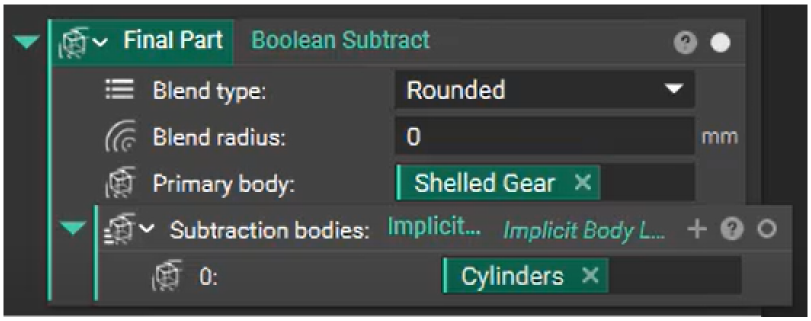
Turn on the visibility of the **Axis** to see that it is already at the desired location.

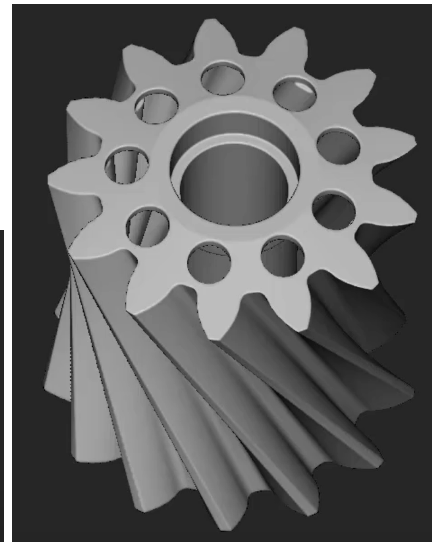




Make the **Polar Array Body** block a variable named ‘Cylinders’.

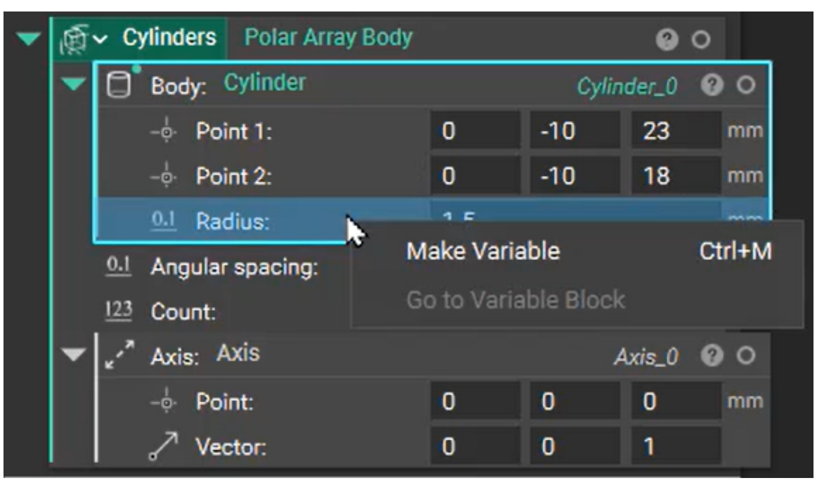
Step 7: Use a **Boolean Subtract** block to subtract the Cylinder bodies from the Shelled Gear. Input 'Shelled Gear' variable as the Primary Body and input the 'Cylinders' variable as the Subtraction Body. Make the **Boolean Subtract** block a variable called 'Final Part'

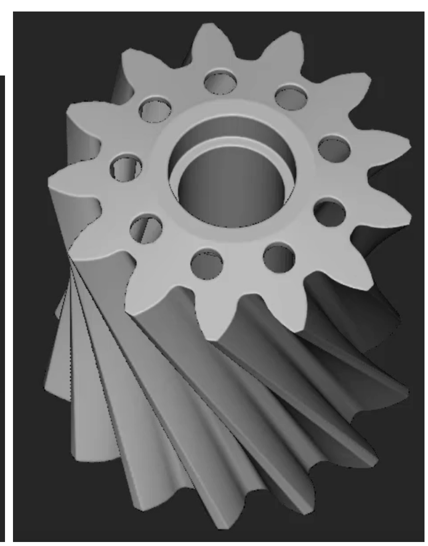




Check to make sure that only the cylinders are only taking away material at the locations of the holes. If this is not the case, continue to adjust Point 1 and Point 2 of the initial **Cylinder** or decrease its radius.

Step 8: Make the Radius input of the Cylinder block a variable named 'Radius of Holes' to be able to quickly find and change it even if the 'Cylinders' variable block is collapsed.





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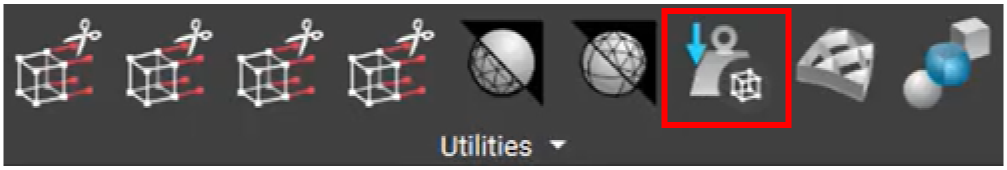
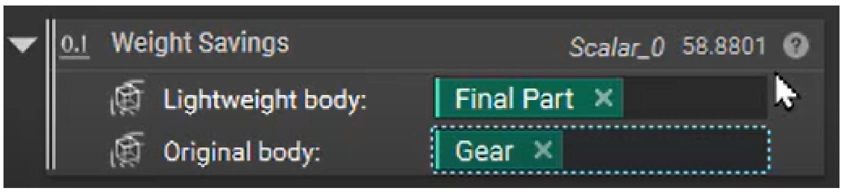
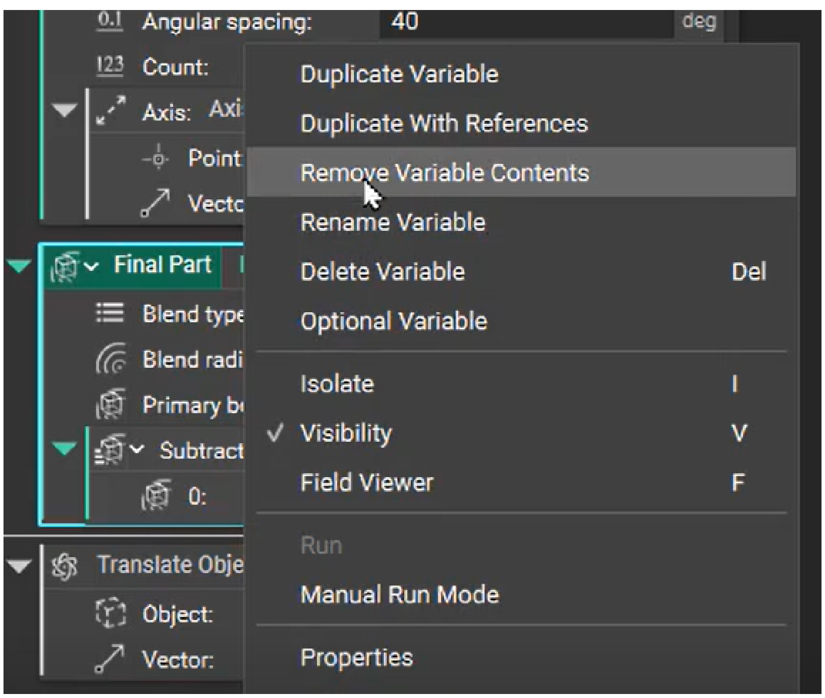
Step 9: Calculate the weight savings using a toolkit block called **Weight Savings**, found in the Lightweighting Tab. The result in percentage shows up on the top right corner of the block.



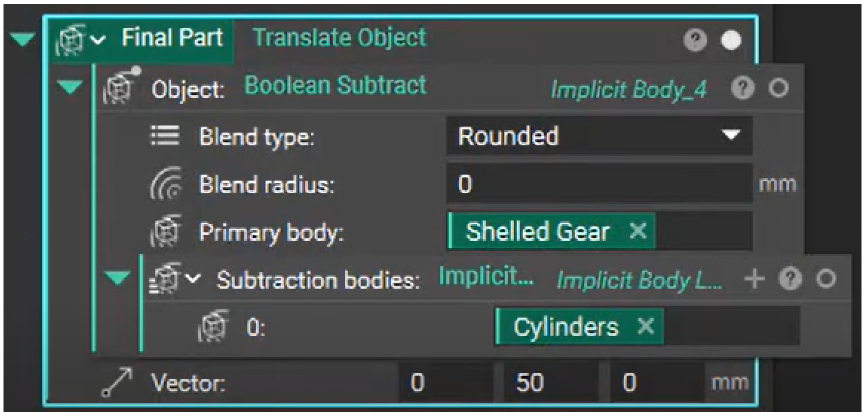


Step 10: Use a **Translate Object** block, found in the Utilities Tab under Transform, to move the 'Final Part' in one direction so that the before and after can be seen together side-by side.

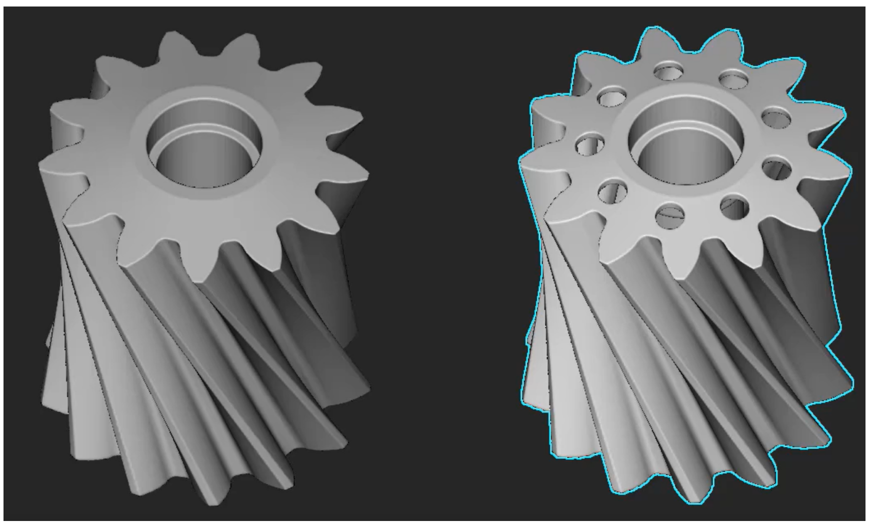
Right-click on the 'Final Part' block and choose 'Remove Variable Contents'.



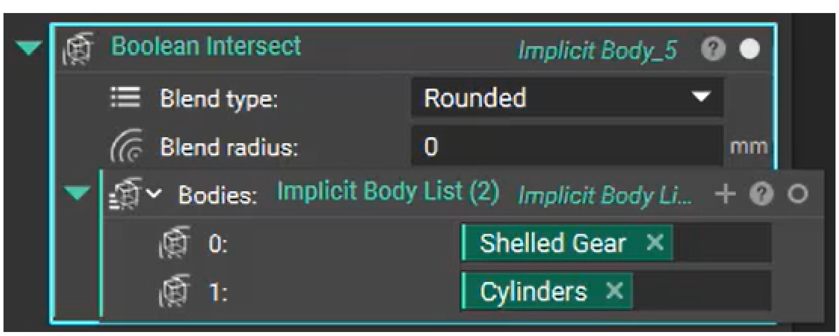
Place the content in the Object input of the **Translate Object** block and the vector value of 50 mm in the y-direction. Then place the filled Translated Object block back into the 'Final Part' Variable.

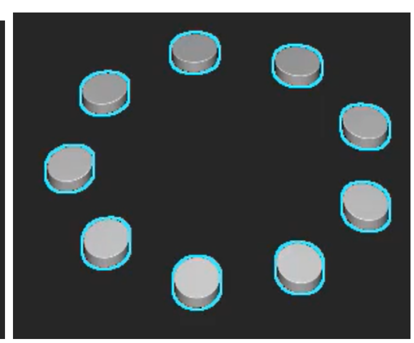


Turn on the visibility of the original 'Gear' block to see both parts on the screen.



Step 11: Create fillings for the holes by using the **Boolean Intersect** block to intersect 'Shelled Gear' and 'Cylinders'. This will keep only the parts that overlap.





Step 12: Translate the **Boolean Intersect** block or the hole fillings to the location of the translated 'Final Part'.

Add a new Translate Object block to the Notebook. Drag the **Boolean Intersect** block into the Object input. For the Vector, go to the 'Final Part' block and make its Vector parameter into a variable named 'Translate Vector'. Use this as the Vector input. Make this new **Translate Object** block a variable called 'Hole Fillings'.