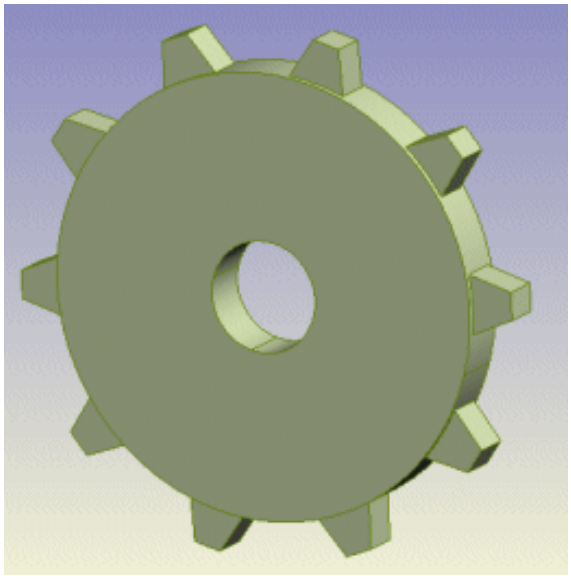
Gear Tutorial

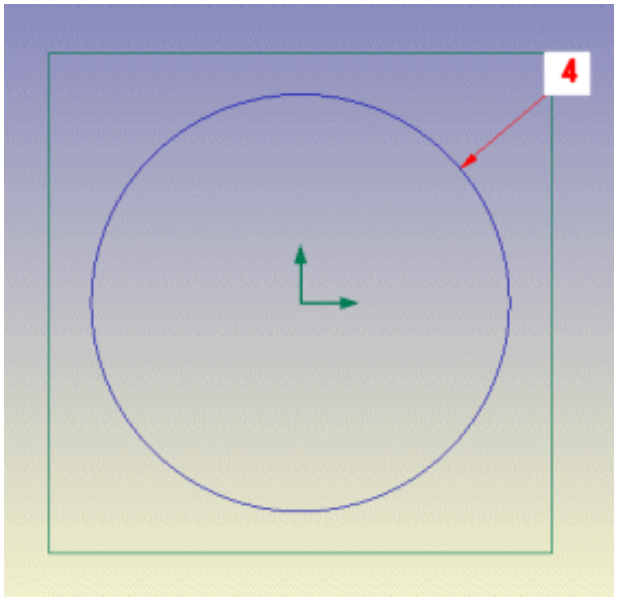


This tutorial will explain how to create a simple gear in Pro/Desktop. You will create a simplified gear with straight teeth. When your are finished you should be able to rapidly modify this gear to make any size with any number of teeth. This should also serve as a foundation for creating different styles and types of gears.

If you are having problems you can get a gear completed using this tutorial.

Start by creating a **File>New>Design**

The gear will be drawn in inches so: **Tools>Options>Units>Inches**



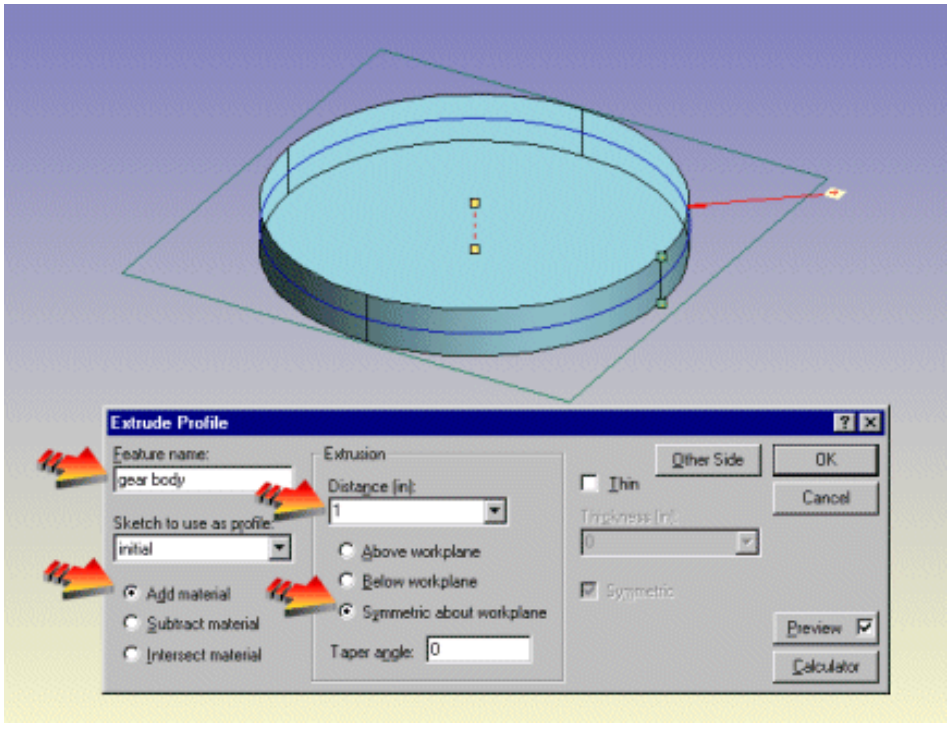
Create a sketch with a simple circle with a RADIUS of 4 inches.

This will be the main gear body

Extrude the circle 

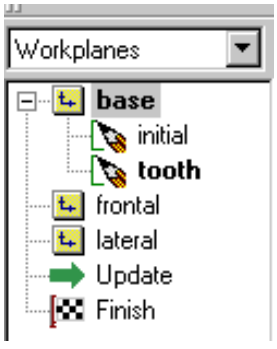
- Feature Name: gear body
- Add Material
- Distance 1 inch
- SYMMETRIC
- OK

The gear is being extruded symmetric so that the teeth can be a different depth that the gear body



Create a new sketch in the base workplane called tooth

- Right Click base in the object browser
- Select New Sketch
- Call the new sketch tooth



View the workplane that the new sketch is on

Draw a two straight line from the center of the circle out beyond the gear body.

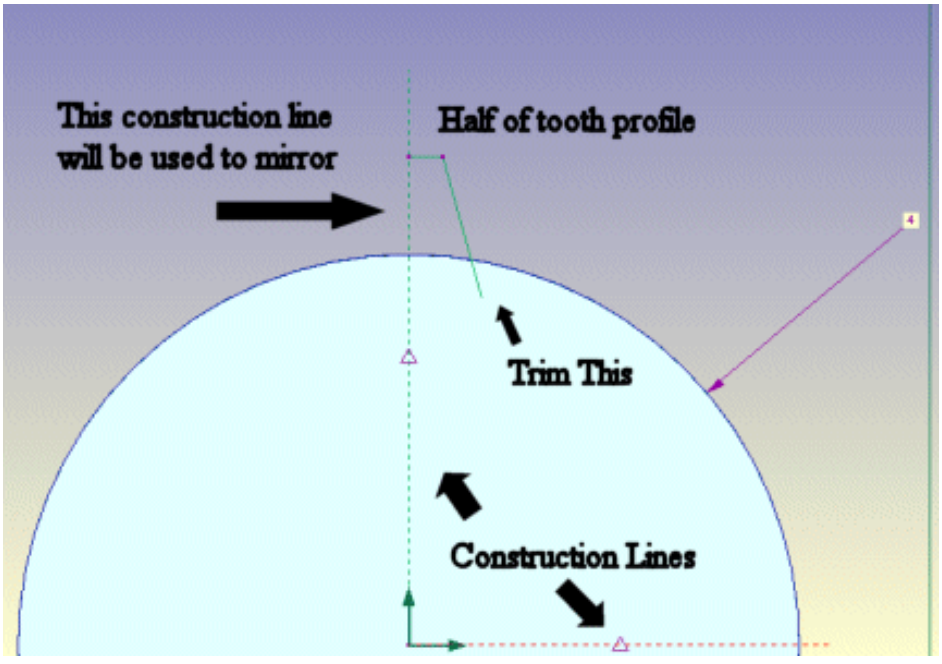
Toggle the lines to construction by: Right Click>Toggle Construction.

Fix each construction line in place by using the fixed constraint button 

Draw a tooth profile as shown. We are only drawing half of the tooth. We

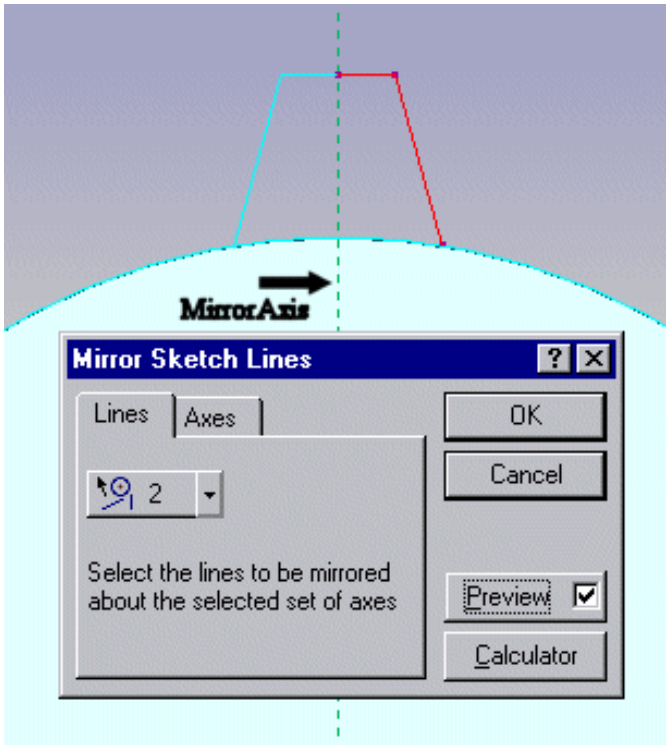
will mirror to get the other half

The side of the tooth is draw thru the gear body to avoid the automatic tangent constriant (line tangent to circle) Use the trim  command to trim off the excess line



After trimming off the excess line you need to mirror the half tooth profile about the construction line.

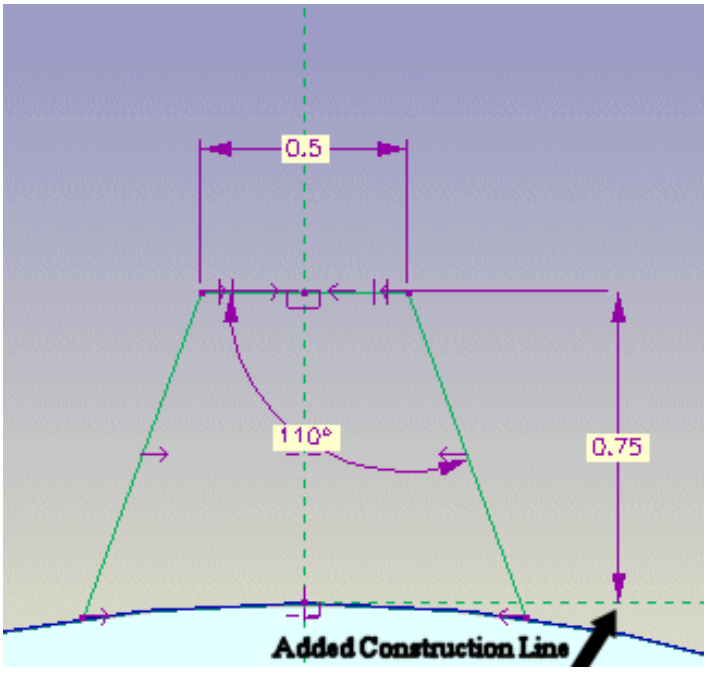
- Select the two lines that make up the profile. Use  and shift select both lines.
- Line>Mirror
- Click on the axis tab and select the construction line
- OK



You will now dimension the tooth

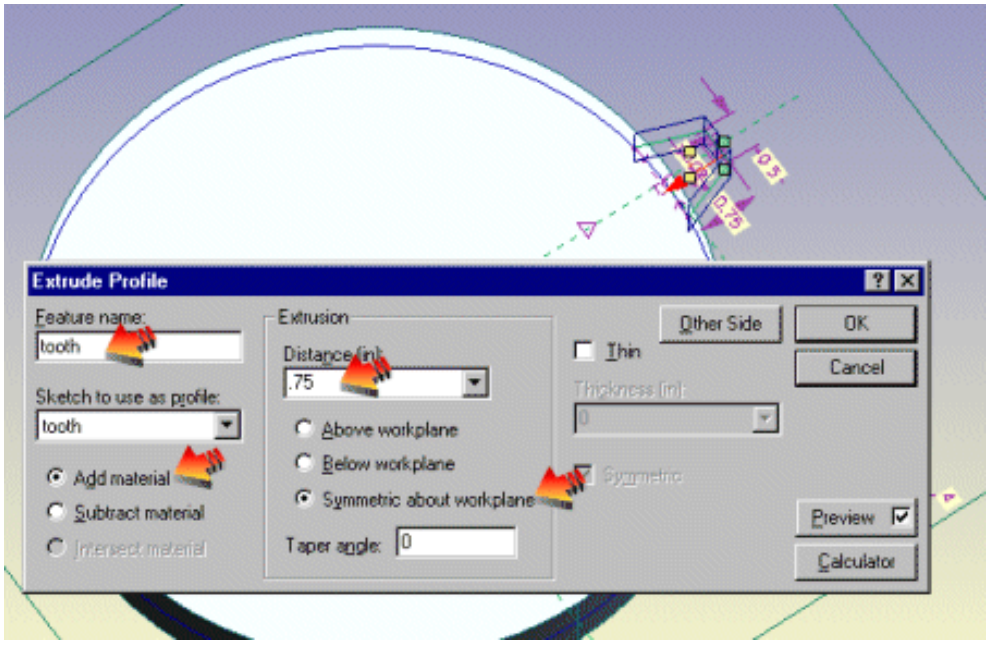
A construction line should be added from where the vertical construction line cross the outer edge of the gear body straight out to the right. This line is only be added for dimensioning purposes. Make sure you make it a construction line.

Add the dimensions shown in the figure to the right. When dimensioning the width (0.5") of the tooth, dimension from point to point. When dimensioning the height of the tooth (0.75"), dimension from the top of the tooth to the newly created construction line.



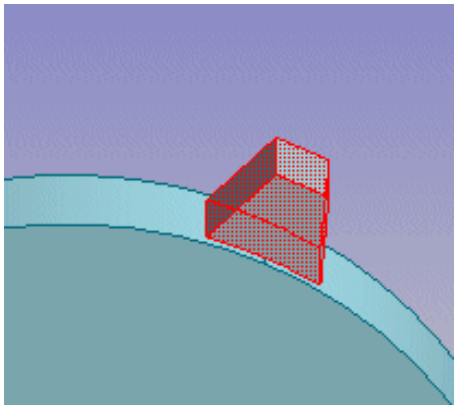
Extrude the tooth 

- Feature Name: Tooth
- Add Material
- Distance .75 inches
- Symmetric
- OK



To create the rest of the teeth you will pattern the tooth feature that you just created,

Use the feature select tool  to select the tooth.



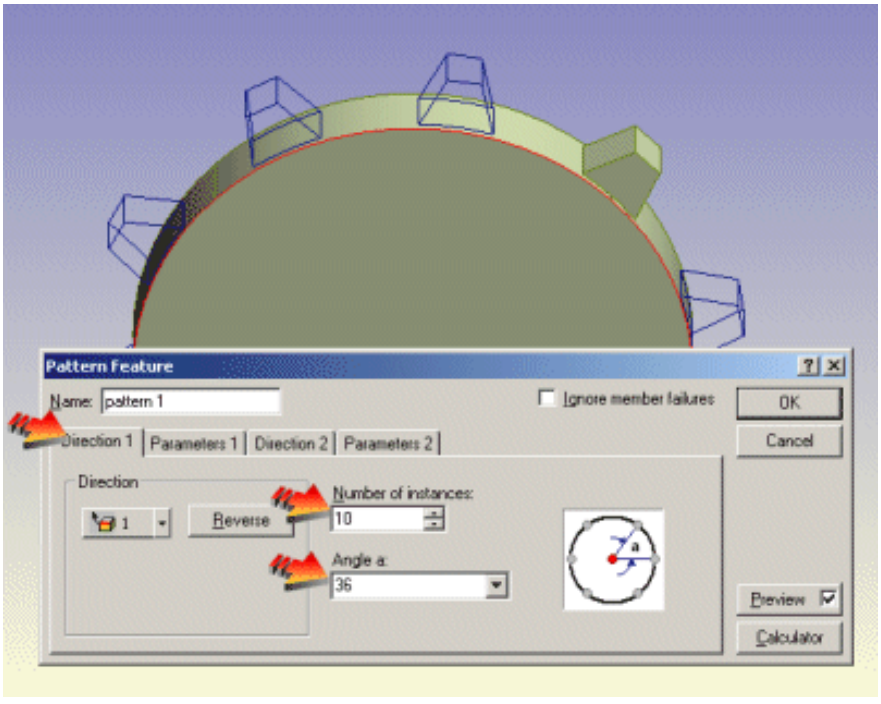
Feature>Pattern

The first thing to do in a pattern is to determine the direction of the pattern

Use the edge select tool  to select the outer edge of the gear body

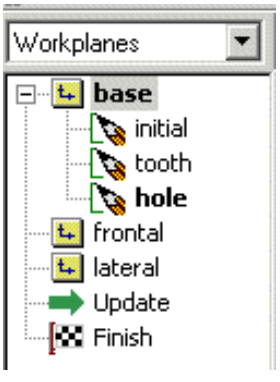
Number of Instances: 10

Angle: 360/10 (this makes the teeth symmetrically distributed)



Finally you will create the hole in the center of the gear body

Right click on the base workplane in the object browser and create a new sketch called hole

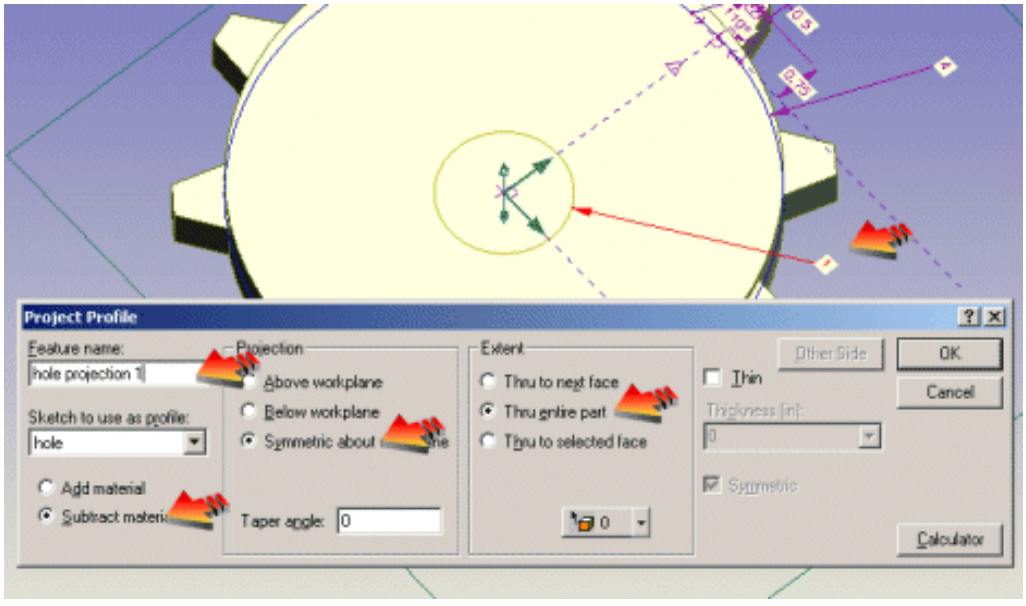


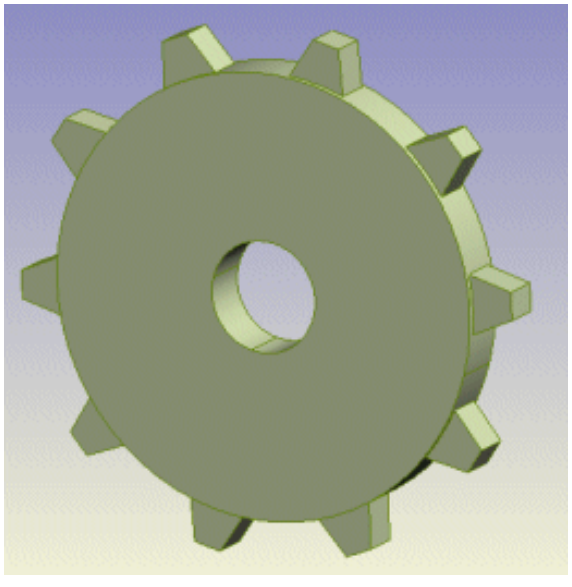
In the hole sketch create a circle concentric to the gear body (in the center) with a RADIUS of 1 inch

Then project the hole:

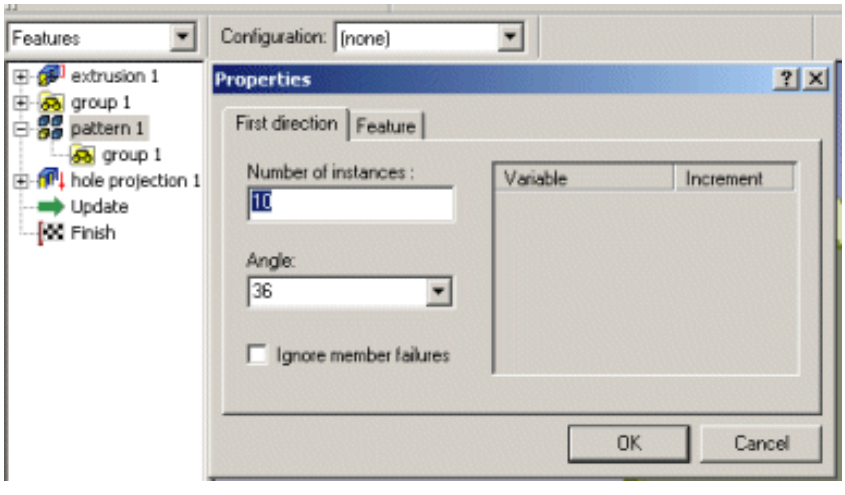
- Subtract Material
- Symmetric
- Thru entire part

By doing the projection symmetric and thru the entire part the hole will always exsist even if the thickness of the gear body is changed





The gear is now complete. You can add more teeth by changing the pattern command. Chabge the object browser to features and double click the pattern feature. Try changing the number of instances and the angles to see what it does



Try changing the body thickness or the tooth thickness.

How about adding rounds to the tips of the tooth and fillets and the base

What happens if you apply a taper to the tooth extrusion?