



How to mate gears in SolidWorks so the tooth flanks are in contact:

In the SolidWorks assembly:

1. Start or use the sketch (GearTeg) at the top of the assembly. This sketch needs to be before the two gears in the assembly feature manager tree.
2. Draw a line that will be between the centers of the gears.
3. Close the sketch.

Note: If your center distance is NOT standard you will need to use the working pitch diameters rather than the standard pitch diameter. This example uses the standard pitch diameters.

On each of the gear part files:

1. Open the auxiliary sketch. Place a point on the pitch diameter circle.
2. Check that it is coincident to the circle.
3. Add a coincident relationship between the point and one side of the involute. The other gear will need to be on the opposite side of the involute. I recommend adding a small circle centered on the points to help see them in the assembly.
4. Close the sketch.

Back in the assembly:

1. Suppress the gear mate, if any.
2. Add a coincident mate to each of the points and the line between the centers of the gears. That should bring each gear into position.
3. Suppress the two new mates and un-suppress or add a gear mate.

To test the mating, an Interference Detection can be run. If no interference is detected, change the center distance slightly (-0.000005") and try again. Also rotate the set of gears and test again.