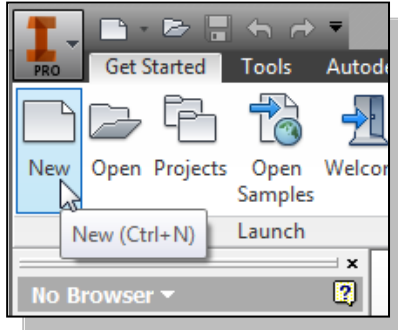# Spur Gears Design

Starting *Autodesk Inventor*

1. Select the **Autodesk Inventor** option on the *Start* menu or select the **Autodesk Inventor** icon on the desktop to start *Autodesk Inventor*. The *Autodesk Inventor* main window will appear on the screen.
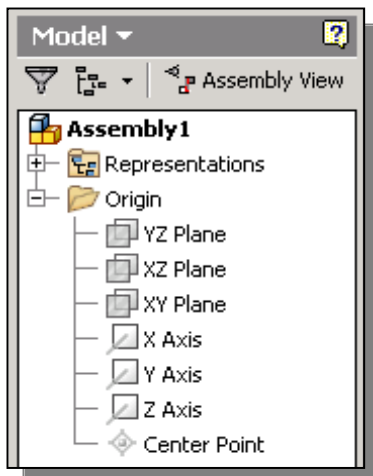
2. Select the **New File** icon with a single click of the left-mouse-button in the *Launch* toolbar as shown.

3. Select the **Metric** tab and in the *Template* list, then select **Standard(mm).iam** (*Standard Inventor Assembly Model* template file).
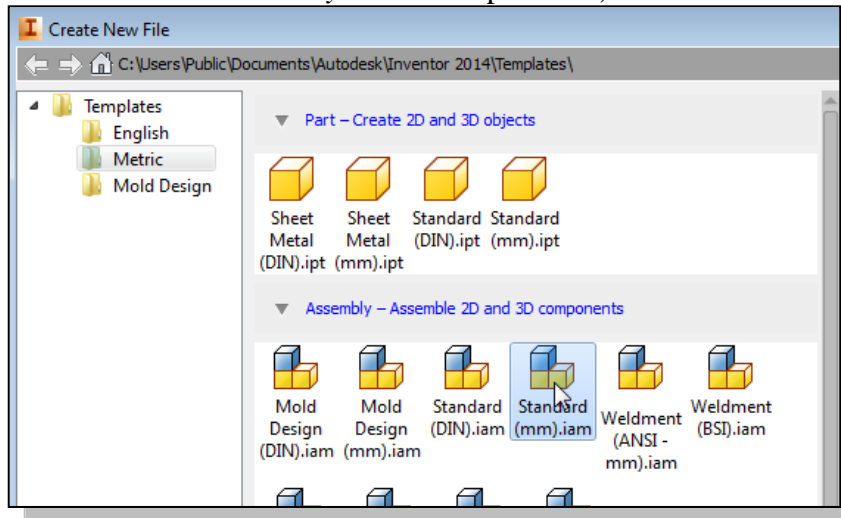
4. Click on the **Create** button in the *New File* dialog box to accept the selected settings and start a new assembly model.

- In the *browser* window, **Assembly1** is displayed with a set of work planes, work axes and a work point. In most aspects, the usage of work planes, work axes and work point is very similar to that of the *Inventor Part Modeler*.

- Notice, in the *Ribbon* toolbar panels, several *component* options are available, such as **Place Component**, **Create Component** and **Place from Content Center**. As the names imply, we can use parts that have been created or create new parts within the *Inventor Assembly Modeler*.

The *Inventor* Spur Gear Generator

- In this section, we will illustrate the procedures to create gears with the Spur Gear component generator. Note that the nylon gears in the *Tamiya Tiger kit* are specifically designed for the *Tamiya kits*, and therefore the tooth profiles do not match with the gear profiles generated using the *Inventor Design Accelerator*.
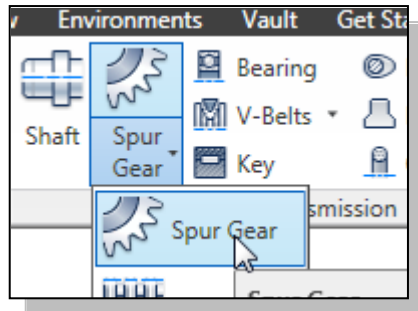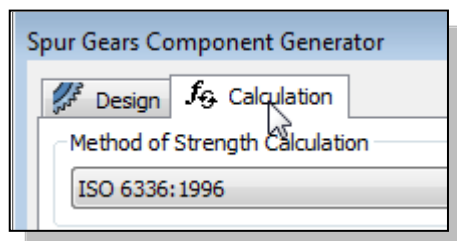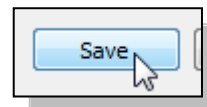
1. In the *Design* tab of the Ribbon toolbar area, select the **Spur Gear** command by left-mouse-clicking the icon.
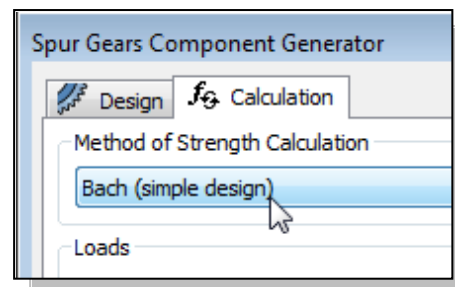
2. Click **OK** to save the current assembly
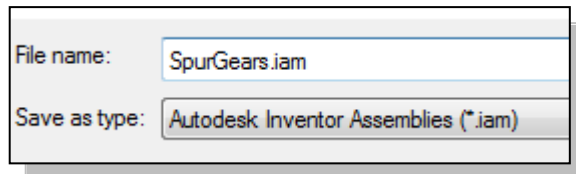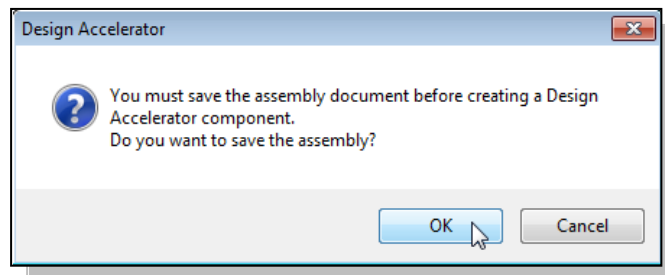
3. Enter **SpurGears** as the assembly model name.

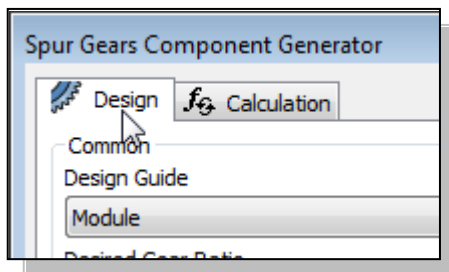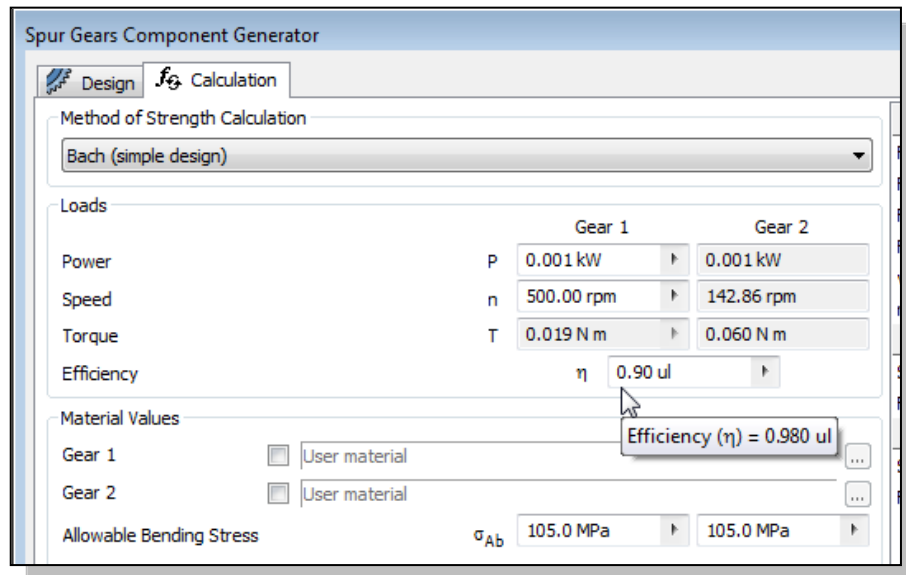4. Click on the **Save** button to save the model file.

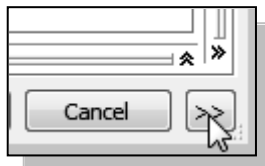5. Click on the **Calculation** tab to set the associated power rating of the system.

6. Choose **Bach (Simple design)** as the method of strength calculation.

7. In the *Loads* section, set the variables using the following values: ***Power*** to **0.001 KW**, ***Speed*** to **500 rpm** and ***Efficiency*** to **0.90** as shown in the figure.
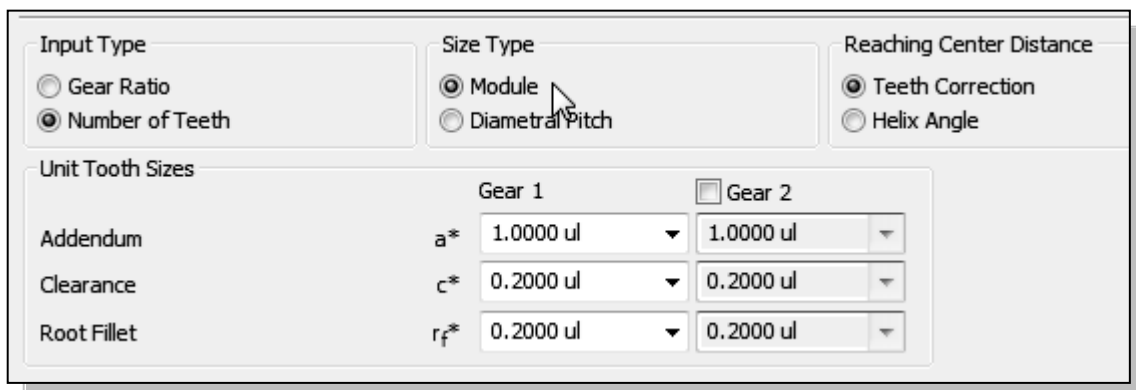
Spur Gears Component Generator

Design | *fG* Calculation

Method of Strength Calculation

Bach (simple design)

Loads

		Gear 1		Gear 2
Power	P	0.001 kW	▸	0.001 kW
Speed	n	500.00 rpm	▸	142.86 rpm
Torque	T	0.019 N m	▸	0.060 N m
Efficiency	η	0.90 ul	▸	

Efficiency (η) = 0.980 ul

Material Values

Gear 1	☐ User material		
Gear 2	☐ User material		
Allowable Bending Stress	σ$_{Ab}$	105.0 MPa ▸	105.0 MPa ▸

Spur Gears Component Generator

Design | *fG* Calculation

Common

Design Guide

Module

8. Click on the **Design** tab to switch back to adjust the settings of the related gears.

Cancel | ≫

9. Click on the **[>>]** button to show the additional settings.

10. Set the *Input Type* to **Number of Teeth** and *Size Type* to **Module** and *Reaching Center Distance* to **Teeth Correction**. Also set the *Unit Tooth Sizes* to the values as shown.

Input Type	Size Type	Reaching Center Distance
○ Gear Ratio	◉ Module	◉ Teeth Correction
◉ Number of Teeth	○ Diametral Pitch	○ Helix Angle

Unit Tooth Sizes

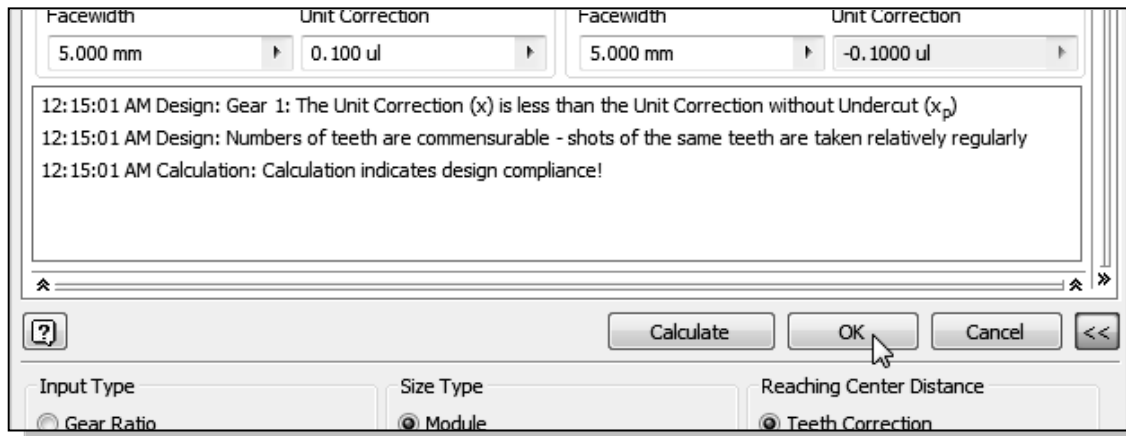|		Gear 1	☐ Gear 2
Addendum	a*	1.0000 ul	1.0000 ul
Clearance	c*	0.2000 ul	0.2000 ul
Root Fillet	r$_f$*	0.2000 ul	0.2000 ul

11. Set the *Design Guide* to **Module**, *Center Distance* to **13.5 mm** and *Pressure Angle* to **22.5 degrees**. Set the gear option to **Component** and enter **12** for the *Number of Teeth* of *Gear 1* and **42** for the *Number Of Teeth* of *Gear 2*. Also set the *Facewidth* to **5 mm** and *Unit Correction* to **0.1**.
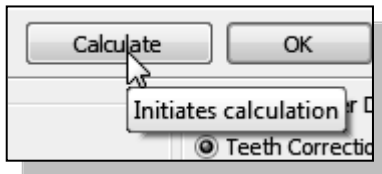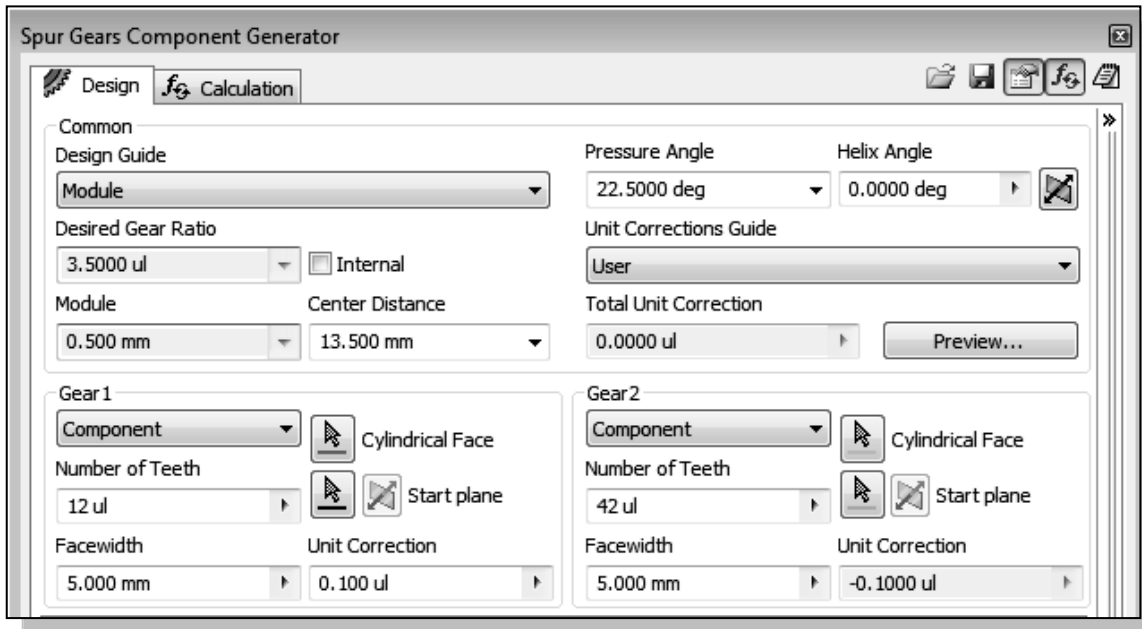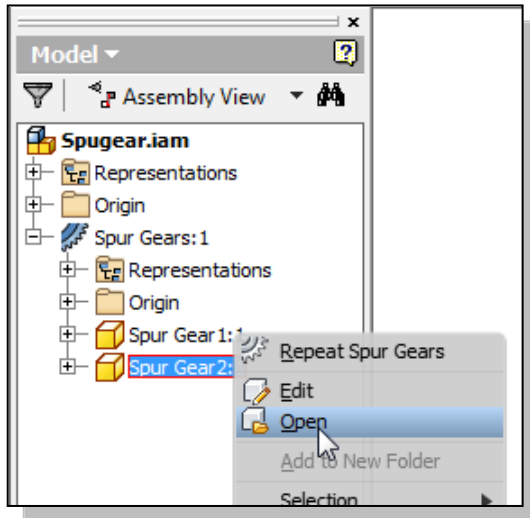

12. Click **Calculation** to initiate the necessary calculations to determine the gear profiles.

13. The results of the calculations are listed in the dialog box as shown. In this case, *Inventor* indicated that the gears will need to be adjusted for proper operation.

14. Click **OK** to proceed with placing the gears in the assembly model.

15. Click **OK** to accept the default naming of the files for the new gears.

File Naming

Item	Display name	File name
Subassembly	Spur Gears	C:\Inventor-Data\Mechanical-Tiger\SpurGears\Design Accelerator\Spur Ge...
Gear	Spur Gear1	C:\Inventor-Data\Mechanical-Tiger\SpurGears\Design Accelerator\Spur Ge...
Gear	Spur Gear2	C:\Inventor-Data\Mechanical-Tiger\SpurGears\Design Accelerator\Spur Ge...

☑ Always prompt for filename   OK   Cancel

16. Place the gears in the assembly model by clicking in an empty location inside the graphics window.

• The *Autodesk Inventor Design Accelerator* provides a convenient way to create a gear set. Note the creation of true gear tooth profiles can be very time consuming by traditional methods. The created gears can also be edited and modified just as a regular *Inventor* part. In the next sections, we will modify these two gears to create the parts needed for the *Tamiya Tiger kit*.

Modify the Generated Gears

1. Inside the *browser* window, **right-click once** on the **Spur Gear 2** to bring up the option menu.

2. In the option menu, choose **Open** to open the *Spur Gear 2* model.

3. On your own, examine the *Spur Gear 2* model. Note the different features used to create the model.

4. Inside the *browser* window, right-click once on the **Extrusion1** to bring up the option menu.

5. In the option menu, choose **Edit Feature** to open the selected feature.
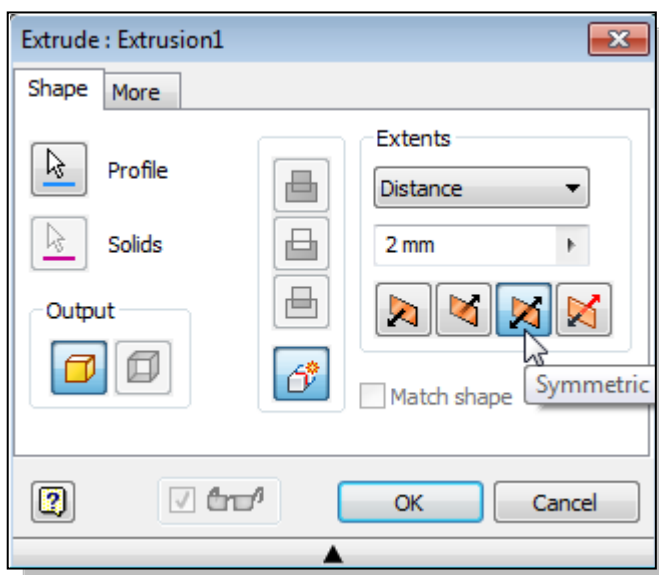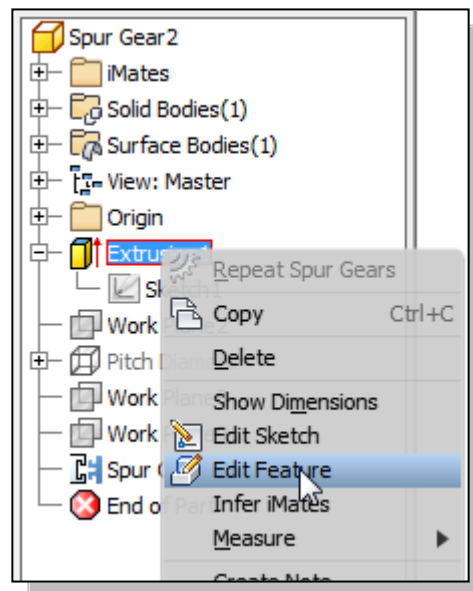
6. Set the extrude distance to **2 mm** and the extrude direction to **Symmetric**. This will set the datum plane in the middle of the solid feature.

7. Click **OK** to accept the settings and end the Edit Feature command.
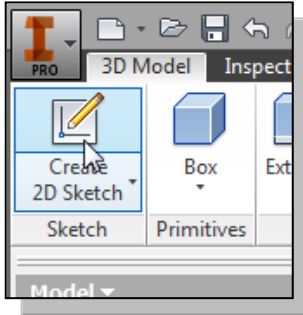
8. In the *Sketch* panel, select the **Create 2D Sketch** command by left-mouse-clicking once on the icon.

9. In the graphics window, select the **top plane** as the sketching plane.
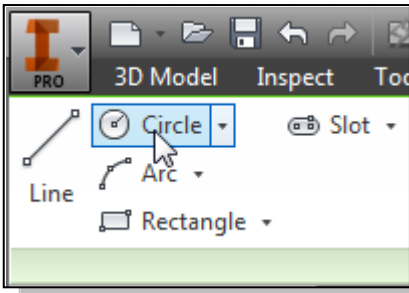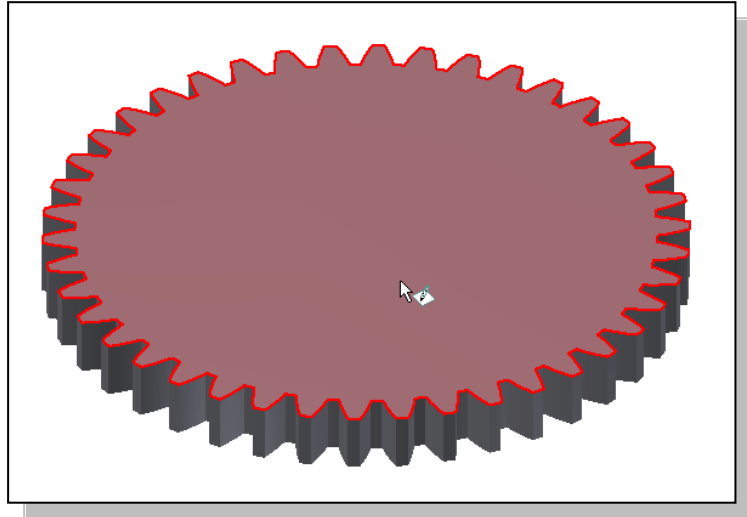
10. Select the **Center point circle** command by clicking once with the left-mouse-button on the icon in the *Draw* toolbar.

11. On your own, create a circle and the associated dimension, diameter of **18 mm**, as shown in figure.
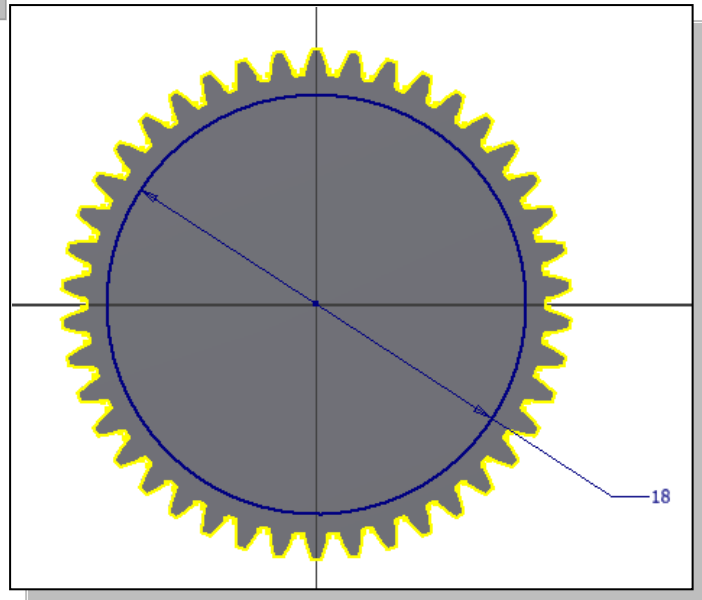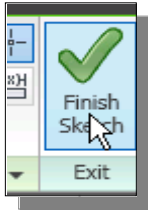
12. Select **Finish Sketch** in the *Ribbon* toolbar to exit the Sketch mode.
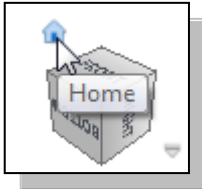
13. In the *View Cube* area, click on the **Home** icon to reset the display to the isometric view.

Complete the Solid Feature
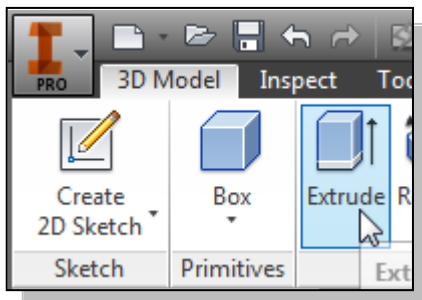
1. In the *Create* toolbar, select the **Extrude** command by releasing the left-mouse-button on the icon.

2. Select the inside region of the 2D sketch to create a profile as shown.

3. Set the extrude option to **Cut** and set the cut distance to **0.25 mm** as shown.
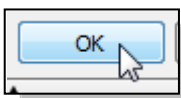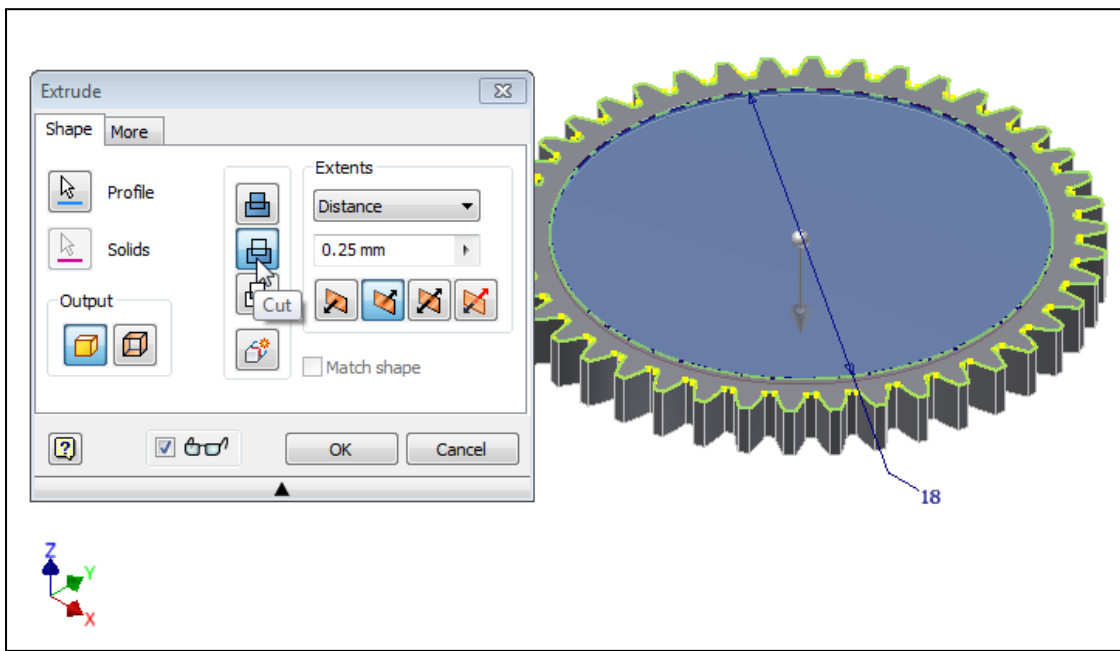
4. Click **OK** to create the cut feature as shown.

Create a Mirrored Feature

❖ The Mirror command can be used to create duplicates of existing features, with respect to a mirror image plane.
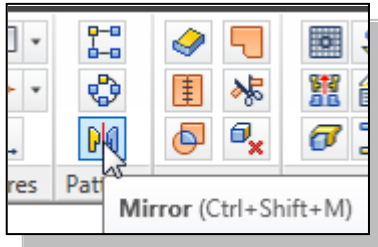
1. In the *Pattern* panel, select the **Mirror Image** command as shown.

2. In the *browser* window, select the **cut extrusion** we just created.
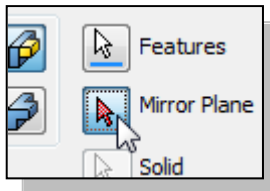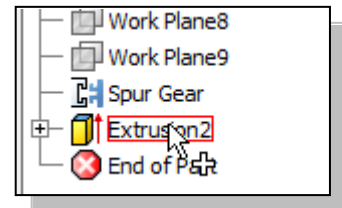
3. Click on the **Mirror Plane** icon, in the *Mirror* dialog box as shown.

4. In the *browser* window, select the **XY Plane** as the mirror image plane as shown.
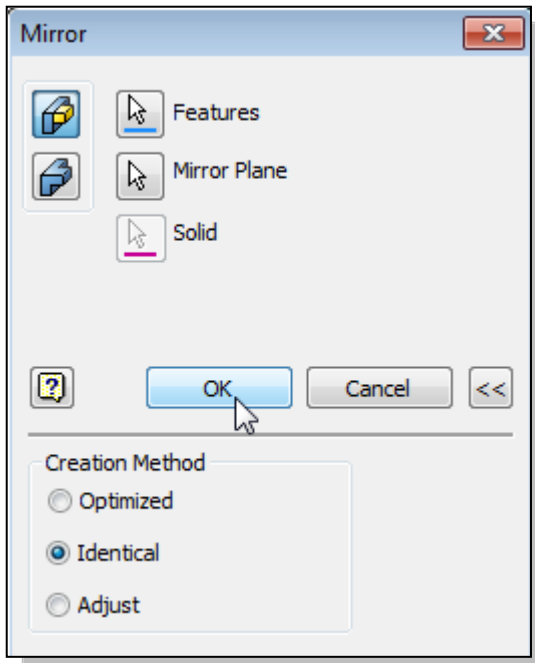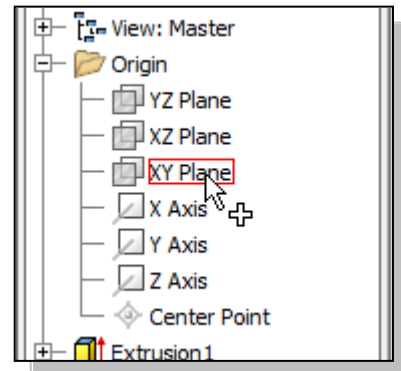
5. In the *Mirror* dialog box, click on the **[<<]** button to view the additional settings as shown.

❖ Note that three options are available. The mirrored duplicate can be created as identical to the original feature, or allow some settings to be adjusted.

6. Click **OK** to create the *Mirrored* feature.

Import the Profile of the Pinion Gear

- For the next feature, we will import the profile of the gear teeth of the pinion gear from the generated gear assembly.
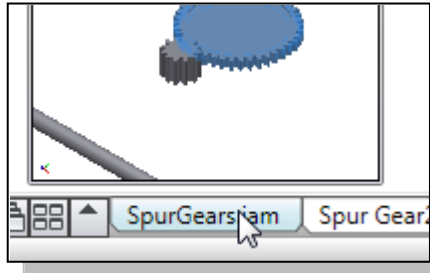
1. In the graphics window, near the bottom edge, select the **SpurGears.iam** tab as shown.
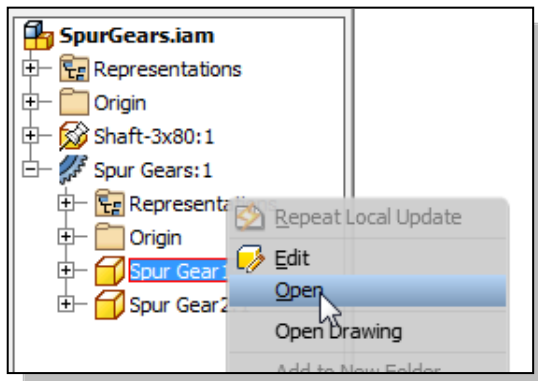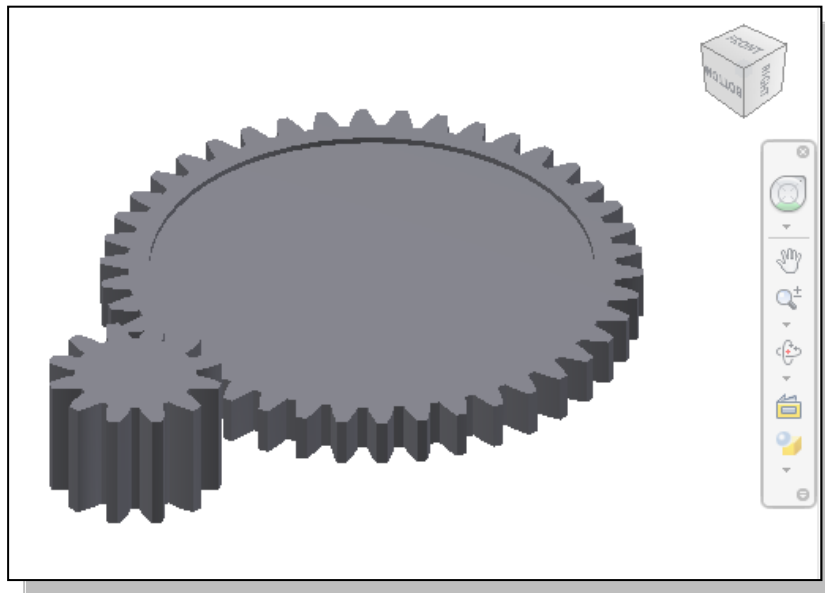
2. Inside the *browser* window, right-mouse-click on **Spur Gears:1** to bring up the option menu as shown.

3. In the option menu, select **Open** to open the model file as shown.
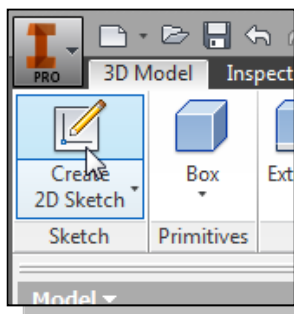
4. In the *Sketch* toolbar, select the **Create 2D Sketch** command by left-clicking once on the icon.

- We will create a copy of the gear teeth profile by using the automatic projected geometry option available in the Sketch mode.
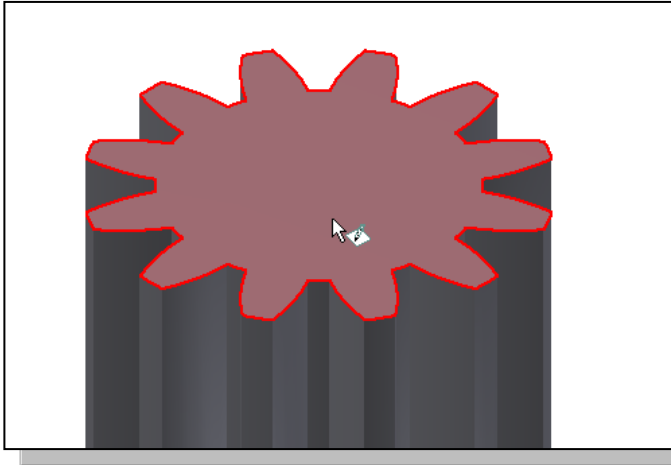
5. Select the **top plane** of the pinion gear as the sketching plane.

6. In the graphics window, use the left-mouse-button to enclose the pinion gear inside a *selection window* as shown.

- By default, *Inventor* Sketch mode will automatically project all of the geometry located on the sketching plane.

Break Link

Repeat Look At

Copy   Ctrl+C

Finish 2D Sketch

Display Degrees of Freedom

Display Curvature

Snap to Grid

Show All Constraints   F8
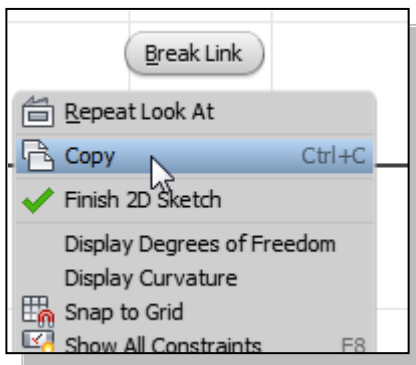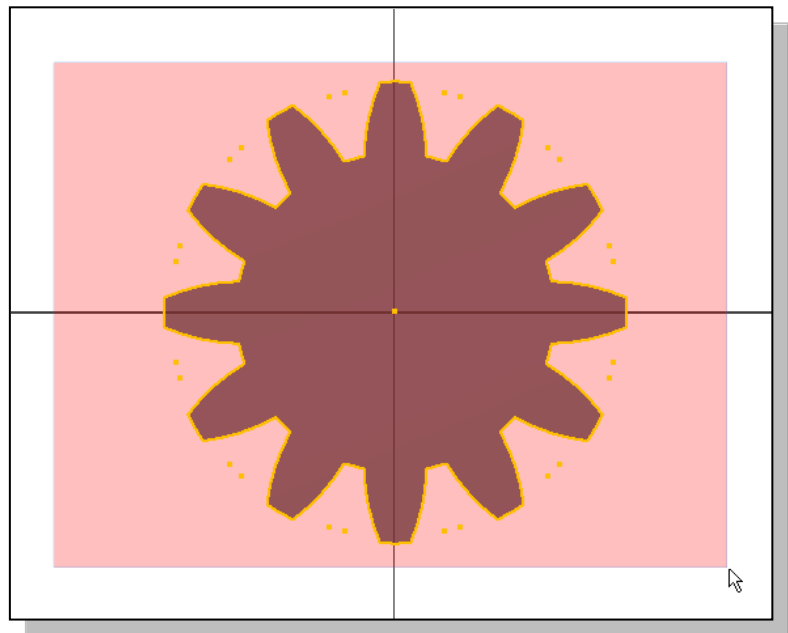
7. Inside the graphics window, right-mouse-click once to bring up the option menu as shown.

8. Select **Copy** as shown. Note that we are using the copy function of the system clipboard.

Spur Gear2.ipt   Spur Ge
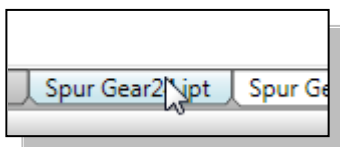
9. In the graphics window, near the bottom edge, select the **SpurGear21.ipt** tab as shown.
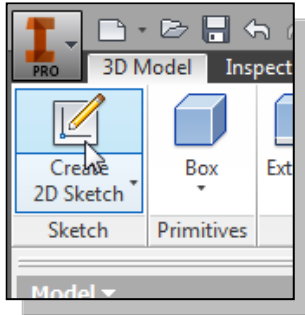
10. In the *Sketch* toolbar, select the **Create 2D Sketch** command by left-clicking once on the icon.
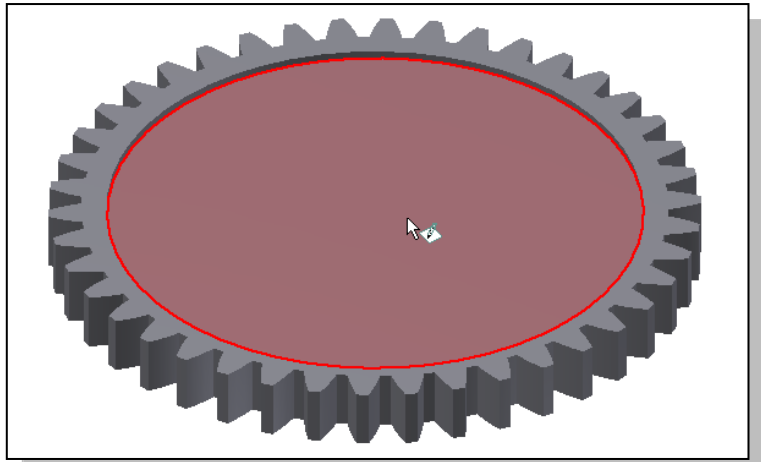
11. Select the **top plane** of the last cut feature as the sketching plane.
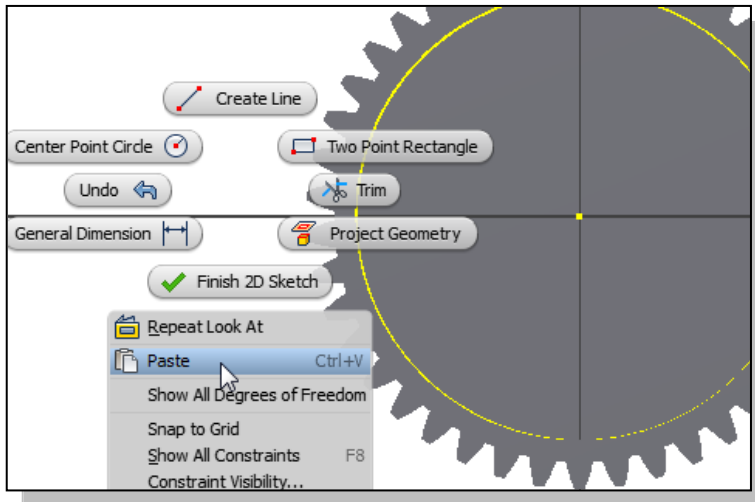
12. Inside the *browser* window, right-mouse-click to bring up the option menu as shown.

13. In the option menu, select **Paste** to paste the profile of the pinion gear as shown.

- Note that the orientation and location of the pasted pinion gear profile is aligned to the *center point* of the coordinate system.
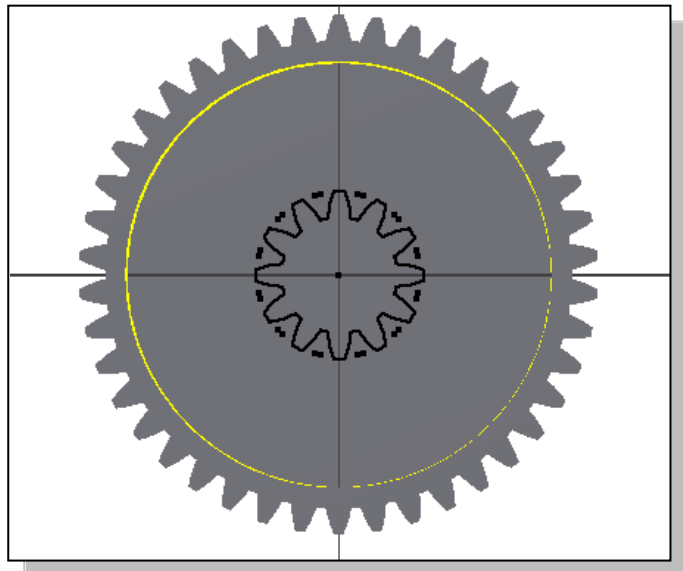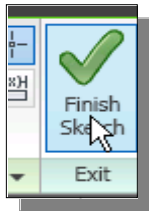
14. Select **Finish Sketch** in the *Ribbon* toolbar to exit the Sketch mode.
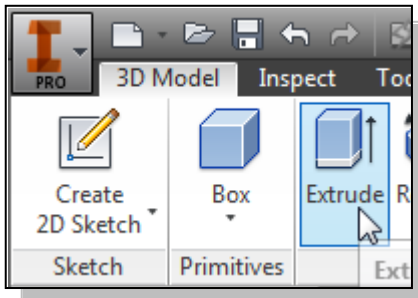
15. In the *Create* toolbar, select the **Extrude** command by clicking the left-mouse-button on the icon.

16. Select the inside region of the 2D sketch to create a profile as shown.

17. Set the extrude distance to **4.75 mm** and create the feature as shown.
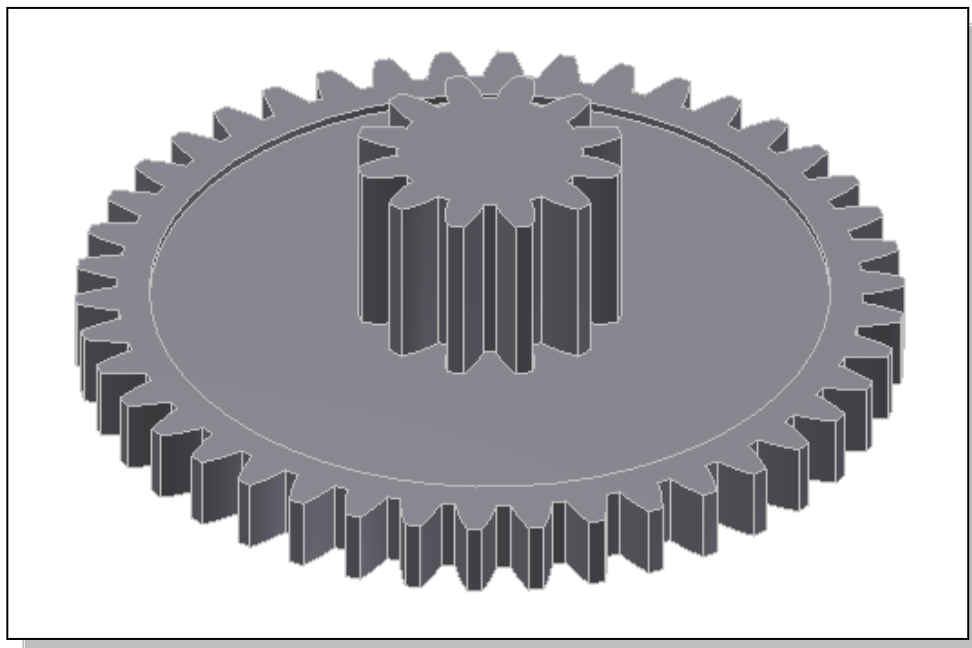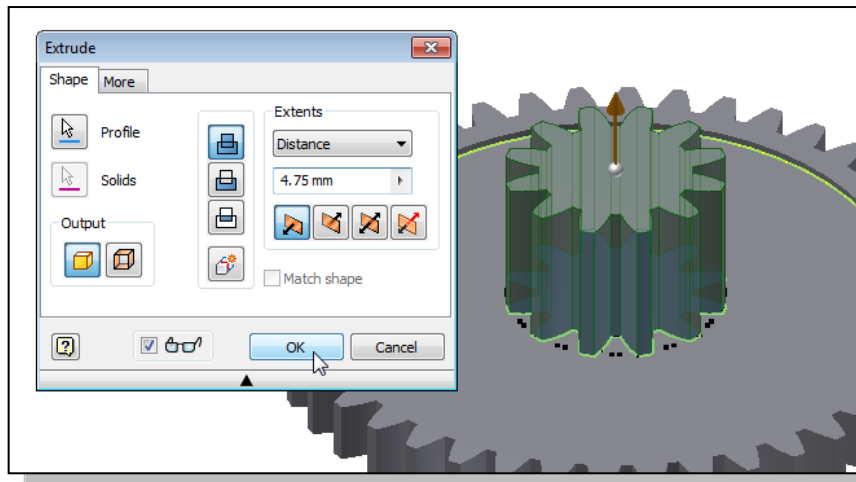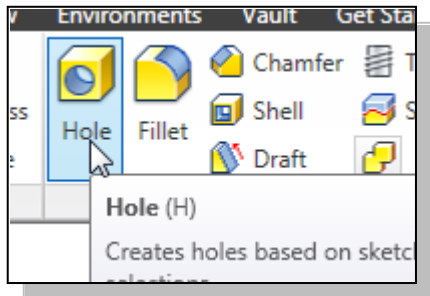
18. In the *Modify* toolbar, select the **Hole** command by clicking the left-mouse-button on the icon.

19. On your own, create a concentric hole feature, diameter **3mm**, aligned to the center of the solid model as shown.
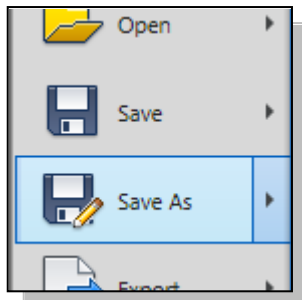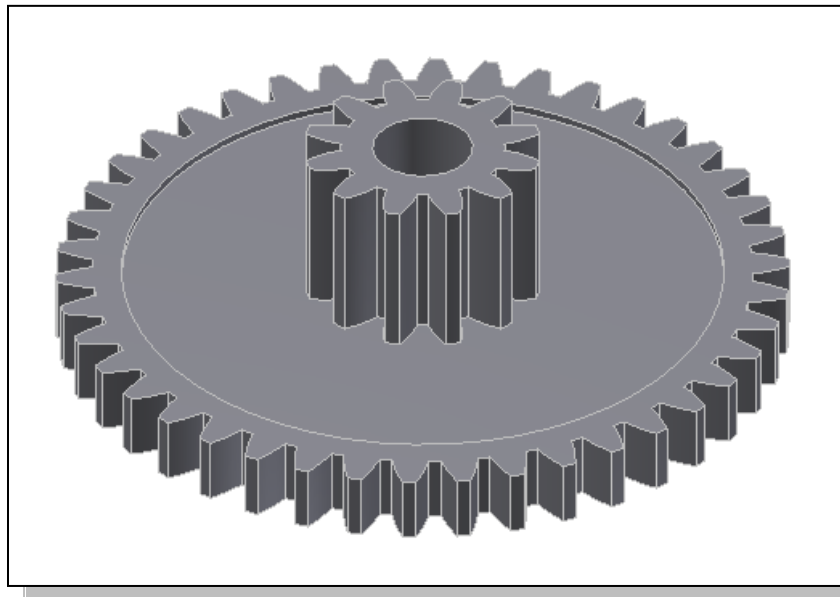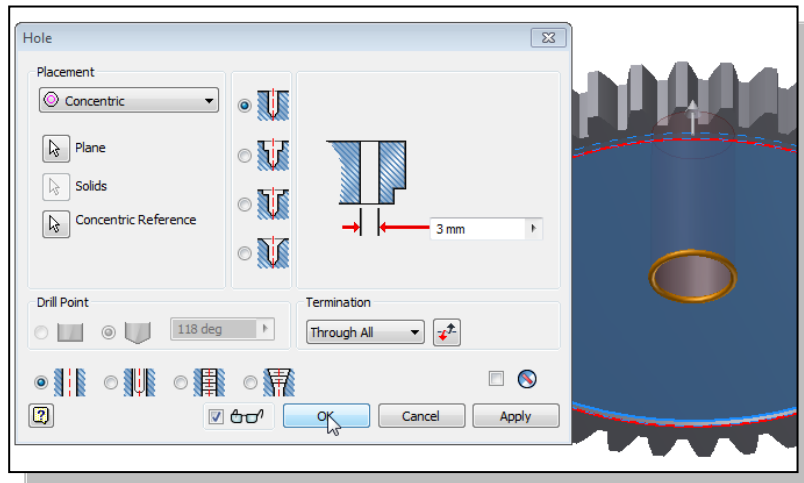
20. On your own, save the current model as a new part using **G2-Spur Gear** as the part file name.

File name:	G2-Spur Gear.ipt
Save as type:	Autodesk Inventor Parts (*.ipt)
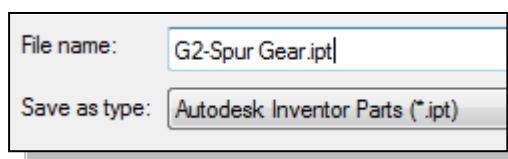
Create the *G3-Spur Gear* Part

❖ Next we will modify the current **Spur Gear 21** part to create the *G3-Spur Gear* part.
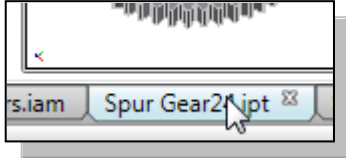
1. Switch to the **Spur Gear 21** part by left-mouse-clicking on the associated tab as shown.
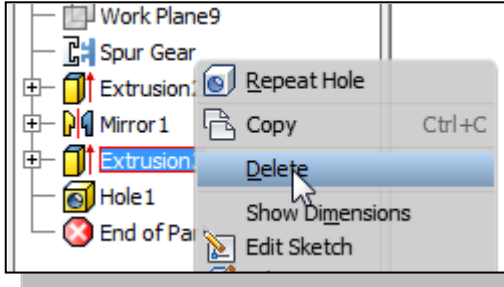
2. Inside the *browser* window, right-mouse-click on **Extrusion3** to bring up the option menu as shown.

3. In the option menu, select **Delete** to remove the extrusion feature as shown.
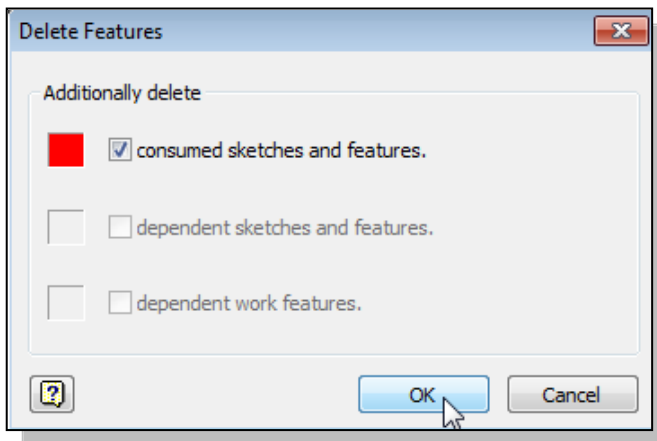
4. Inside the *Delete Features* dialog box, confirm the **delete associated sketches** option is turned *ON*.

5. Click **OK** to accept the settings and delete the selected feature.
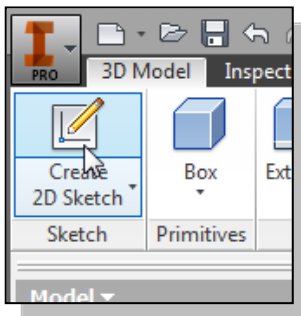
6. In the *Sketch* toolbar, select the **Create 2D Sketch** command by left-clicking once on the icon.

7. Select the **top plane** of the cut feature as the sketching plane as shown.
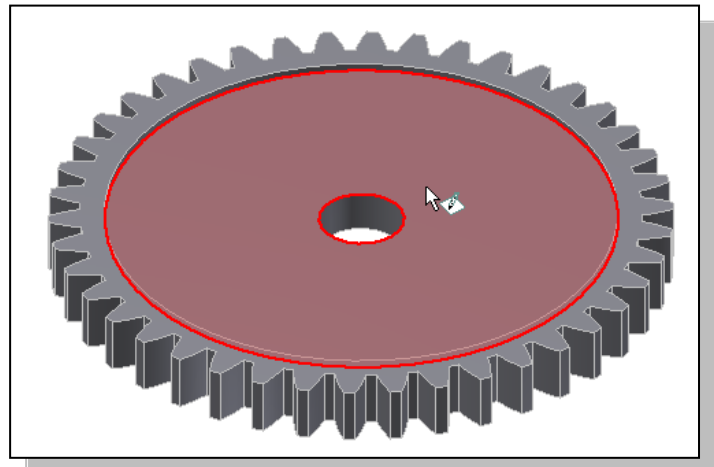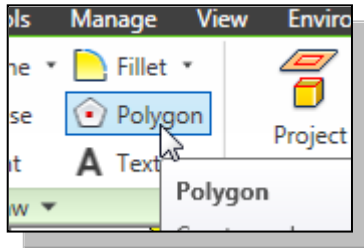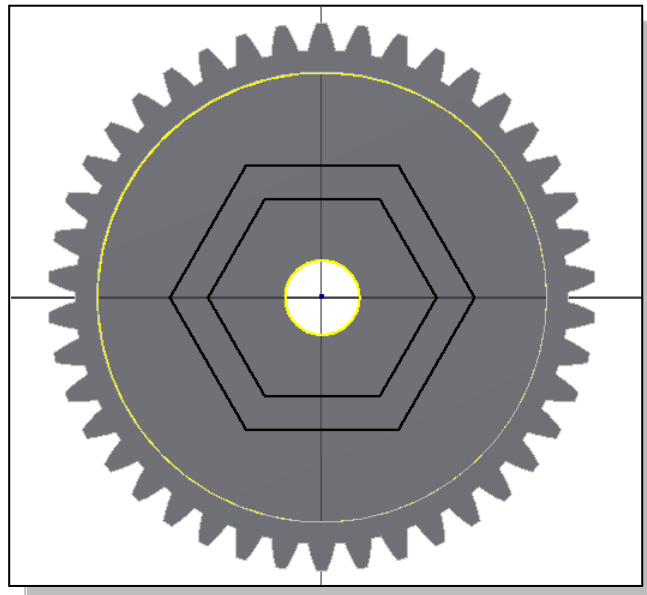
8. In the *Draw* toolbar panel, select the **Polygon** command by left-mouse-clicking once on the icon.
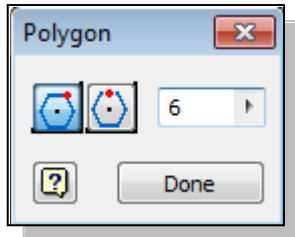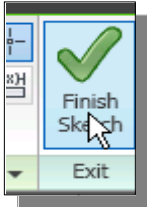

9. Set the polygon option to **Inscribed** and number of sides to **6** in the *Polygon* dialog box as shown.

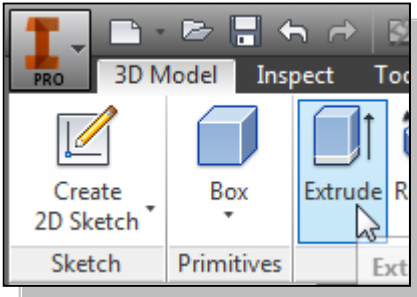10. Create two polygons aligned to the center point of the gear as shown.
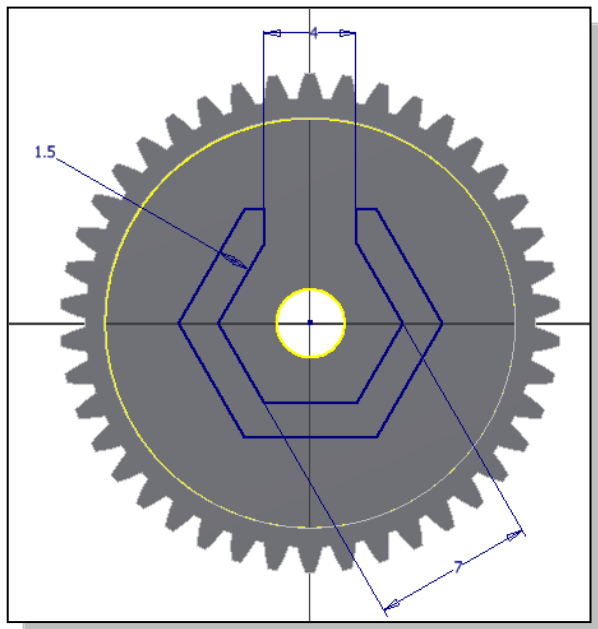
11. On your own, modify and complete the sketch as shown in the figure.

12. On your own, apply the necessary constraints and three dimensions, **7 mm**, **4 mm**, and **1.5 mm** to fully constrain the sketched geometry.

13. Select **Finish Sketch** in the *Ribbon* toolbar to exit the Sketch mode.


14. In the *Create* toolbar, select the **Extrude** command by releasing the left-mouse-button on the icon.

15. Select the inside region of the 2D sketch to create a profile as shown.

16. Set the extrude distance to **2.75 mm** and create the feature as shown.
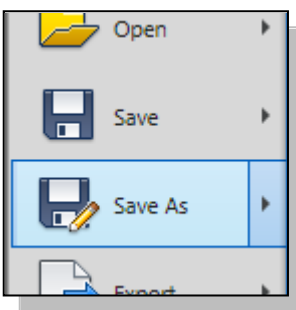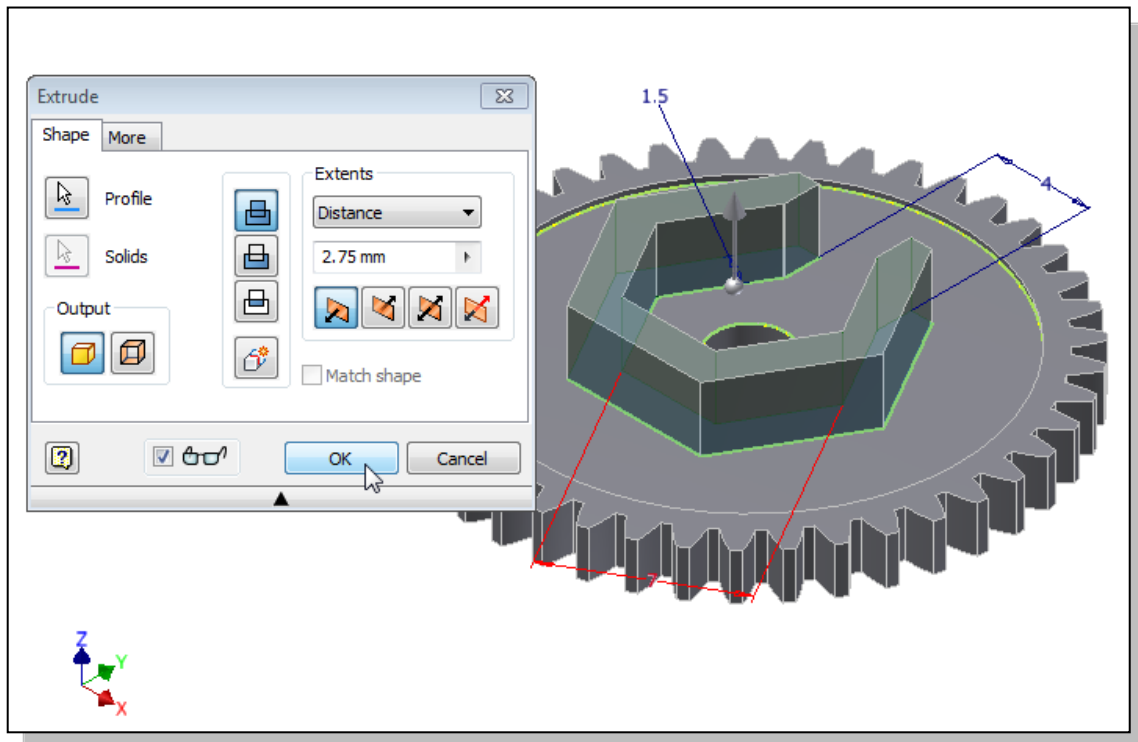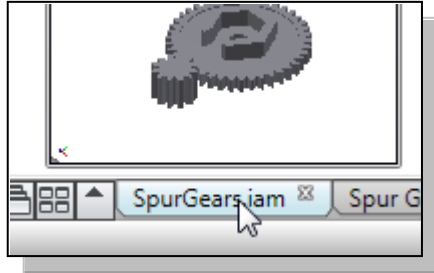

17. On your own, save the current model as a new part using **G3-Spur Gear** as the part file name.

| File name: | G3-Spur Gear.ipt |
| Save as type: | Autodesk Inventor Parts (*.ipt) |
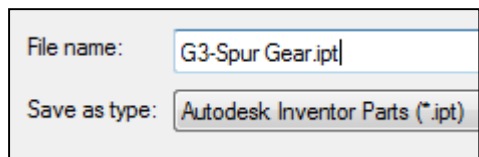
Create another Spur Gear Set

❖ For the two other gears, we will first create another set of spur gears by using the *Design Accelerator*.
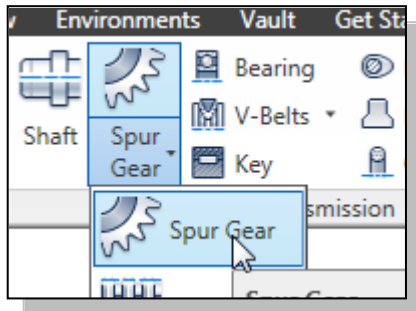

1. In the graphics window, near the bottom edge, select the **SpurGears.iam** tab as shown.
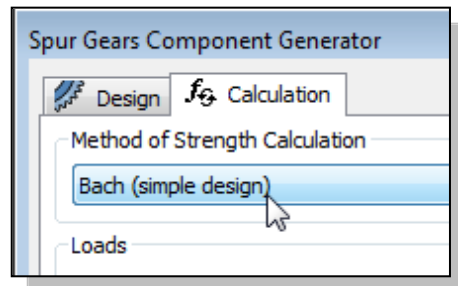

2. In the *Ribbon* panel, select the **Design** tab by left-mouse-clicking the tab.
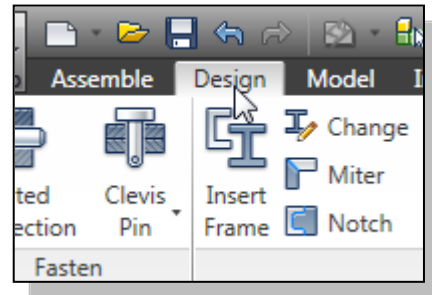
3. In the *Transmission* panel, select the **Spur Gear** command by left-mouse-clicking the icon.
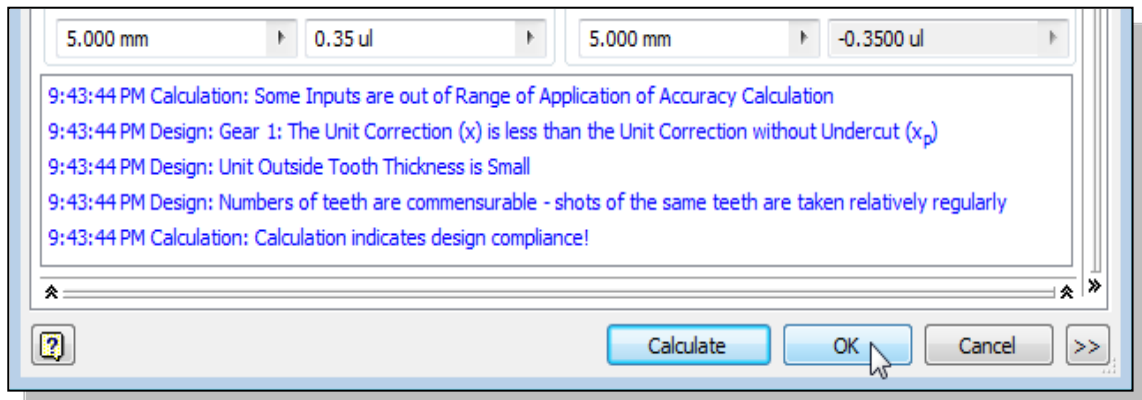

4. Click on the **Calculation** tab to set the associated power rating of the system. Choose **Bach (Simple design)** as the *Method of Strength Calculation*.

5. On your own, set the **8T** and **72T** gears information as shown in the figure.
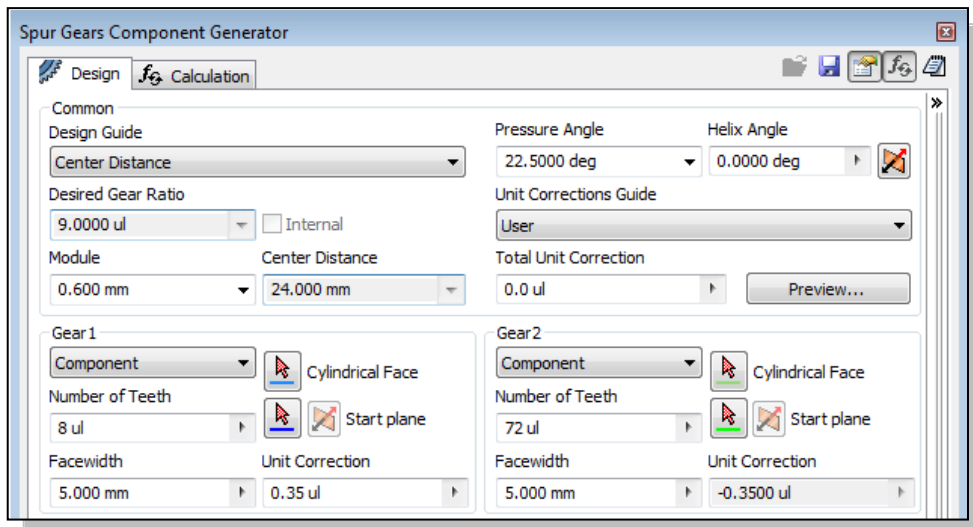
6. The results of the calculations are listed in the dialog box as shown. In this case, *Inventor* indicated that the gears will need to be adjusted for proper operation.
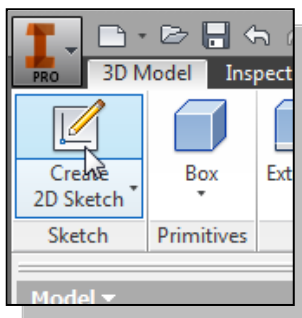
5.000 mm	▸	0.35 ul	▸	5.000 mm	▸	-0.3500 ul	▸

9:43:44 PM Calculation: Some Inputs are out of Range of Application of Accuracy Calculation
9:43:44 PM Design: Gear 1: The Unit Correction (x) is less than the Unit Correction without Undercut (x_p)
9:43:44 PM Design: Unit Outside Tooth Thickness is Small
9:43:44 PM Design: Numbers of teeth are commensurable - shots of the same teeth are taken relatively regularly
9:43:44 PM Calculation: Calculation indicates design compliance!

[Calculate]   [OK]   [Cancel]   [>>]

7. Click **OK** to accept the settings. On your own, place a copy of the generated gears in the assembly model.

Create the G0-Pinion Part
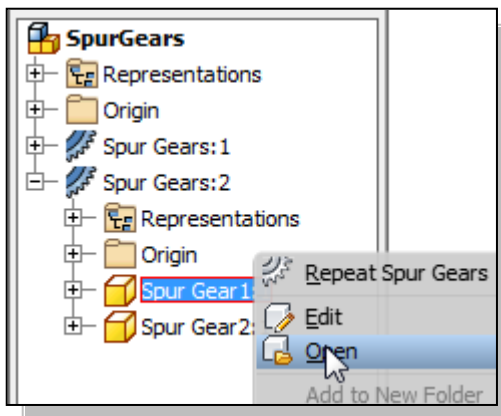
1. Inside the *browser* window, right-mouse-click on the **pinion gear**, on the second set of gears, to bring up the option menu as shown.

2. In the option menu, select **Open** to open the selected model as shown in the figure.

SpurGears
  ⊞ Representations
  ⊞ Origin
  ⊞ Spur Gears:1
  ⊟ Spur Gears:2
      ⊞ Representations
      ⊞ Origin
      ⊞ Spur Gear1
      ⊞ Spur Gear2

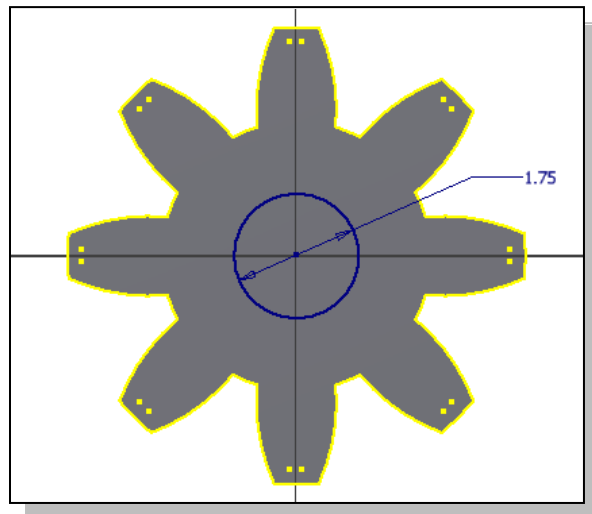Repeat Spur Gears
Edit
Open
Add to New Folder

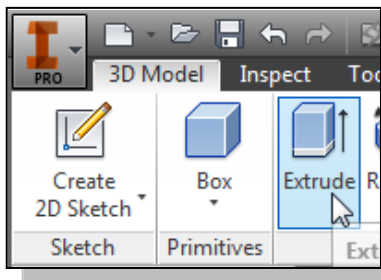3. In the *Sketch* toolbar, select the **Create 2D Sketch** command by left-clicking once on the icon.
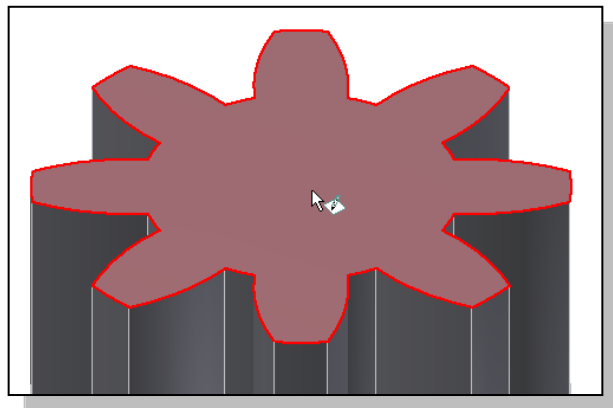
4. Select the **top plane** of the pinion gear model as the sketching plane.

5. On your own, create a diameter **1.75 mm** circle aligned to the center of the pinion gear.

6. Select **Finish Sketch** in the *Ribbon* toolbar to exit the Sketch mode.

7. In the *Create* toolbar, select the **Extrude** command by releasing the left-mouse-button on the icon.

8. Select the inside region of the 2D sketch to create a profile as shown.

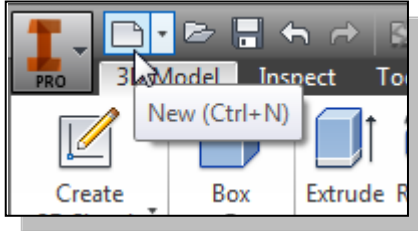9. Set the extrude extents to **All** and extrude option to **Cut** as shown.
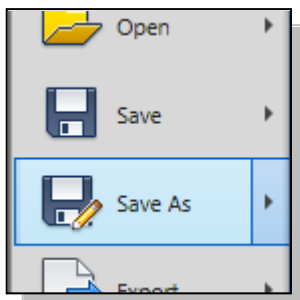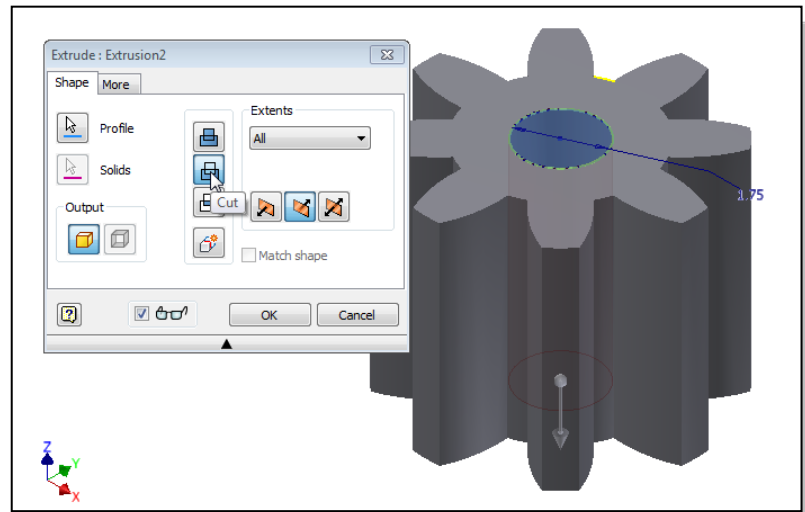
10. On your own, save the current model as a new part using **G0-Pinion** as the part file name.
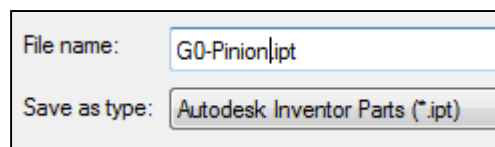
Start a New Part File

❖ The last gear we need for the *Tamiya Tiger kit* is a non-standard compound gear. We will create this gear using several options, with a portion of it using the **Revolve** *Pattern* commands.

1. In the *Quick Access* menu, select **New** to start a new file.

2. In the *New File* dialog box, select the **Standard (mm).ipt** template by left-mouse-clicking once on the icon.
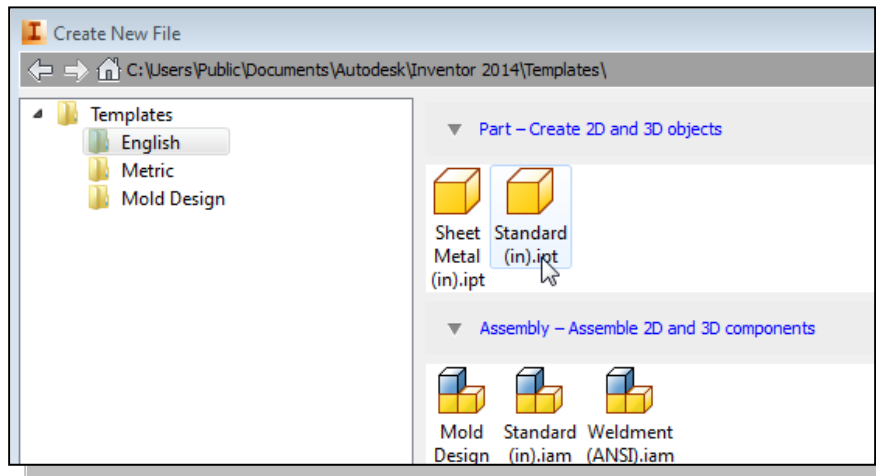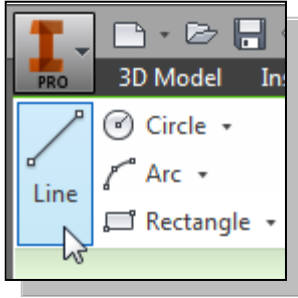

3. Click **Create** to accept the selection and start a new file.
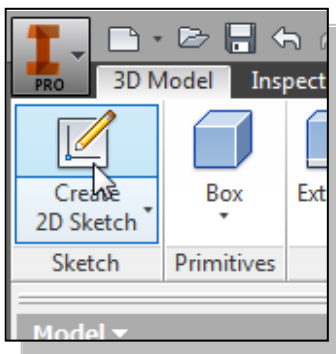
4. In the *Sketch* toolbar, select the **Create 2D Sketch** command by left-clicking once on the icon.

5. Inside the *browser* window, select the XZ Plane as the sketching plane as shown.
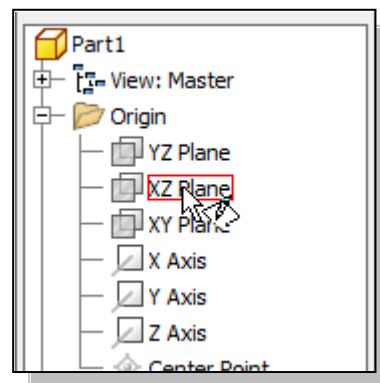
6. Select the **Line** command by clicking once with the left-mouse-button on the icon in the *Draw* toolbar.

7. Create **a closed region** sketch to the right side of the center point; also create **a vertical line** aligned to the center point of the coordinate system as shown.
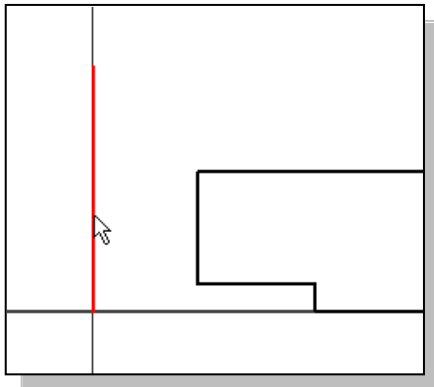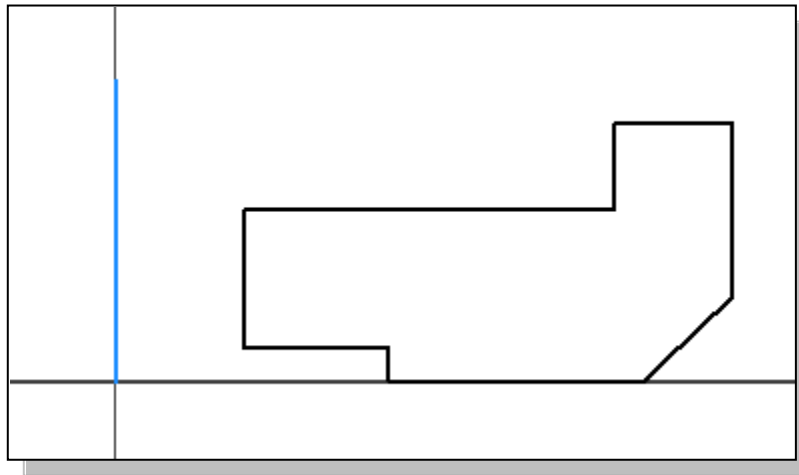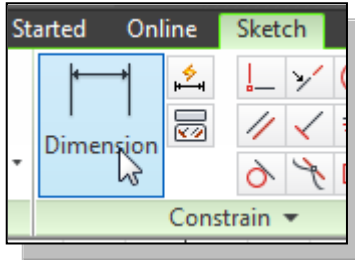

8. Select the vertical line as shown.

❖ For the **Revolve** command, a center axis of rotation is required. We will create a center line by converting the created vertical line.
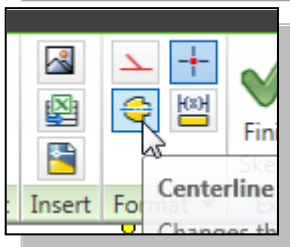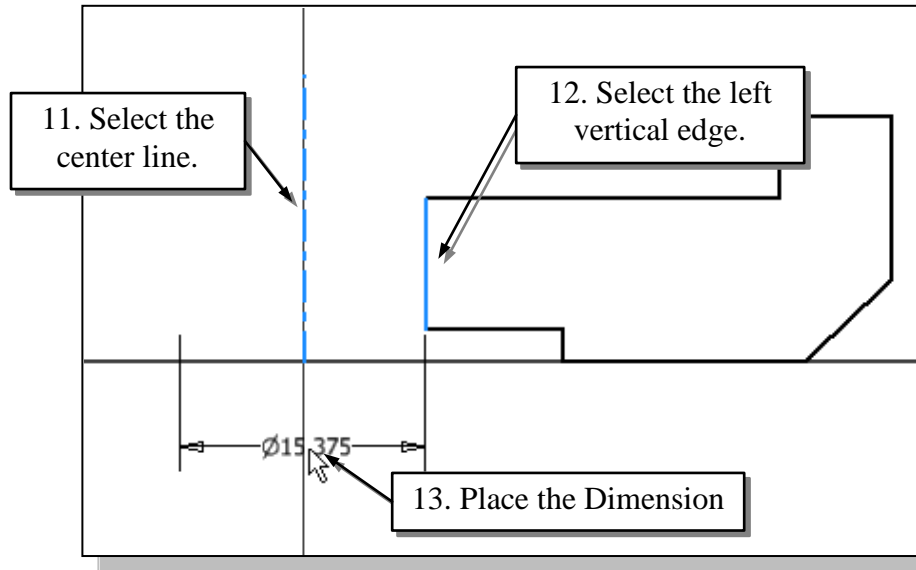
9. In the *Ribbon* toolbar, select the **Centerline** option to convert the created vertical line to a center line as shown.
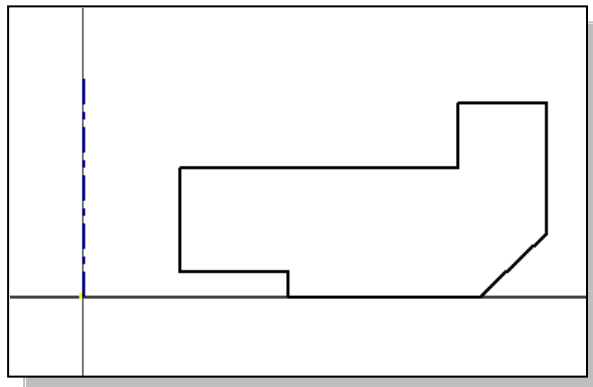
10. Select the **Smart Dimension** command in the *Constrain* panel.

11. Pick the **center line** as the first entity to dimension as shown in the figure.

11. Select the center line.

12. Select the left vertical edge.

Ø15.375

13. Place the Dimension

12. Select the **left vertical line** as the second object to dimension.

13. Place the dimension text below the center point of the coordinate system.

- To create a dimension that will account for the symmetrical nature of the design, pick the axis of symmetry, pick the entity, and then place the dimension. (The **Linear Dimension** option is also available through the right-mouse-click option menu.)
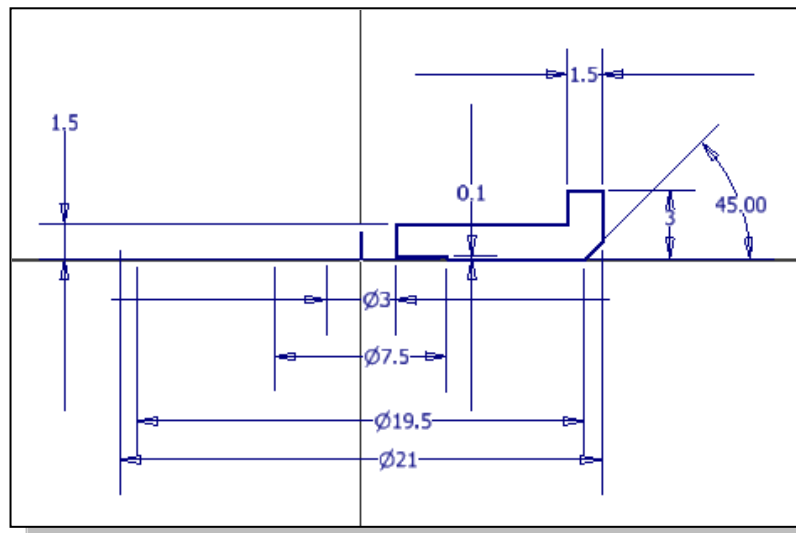
1.5

1,5

0,1
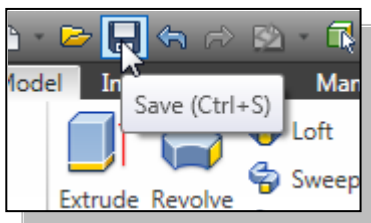
1.5

45.00

Ø3

Ø7.5

Ø19.5

Ø21

14. Select **Finish Sketch** in the *Ribbon* toolbar to exit the Sketch mode.
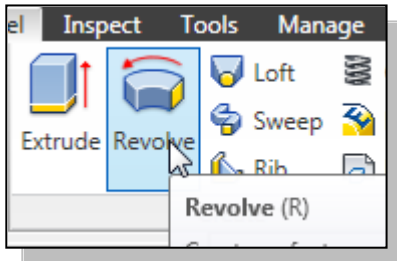
15. In the *Create* toolbar, select the **Revolve** command by clicking the left-mouse-button on the icon. Note the profile and the center axis are automatically selected for the revolved feature.

Revolve (R)
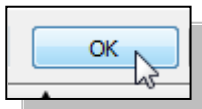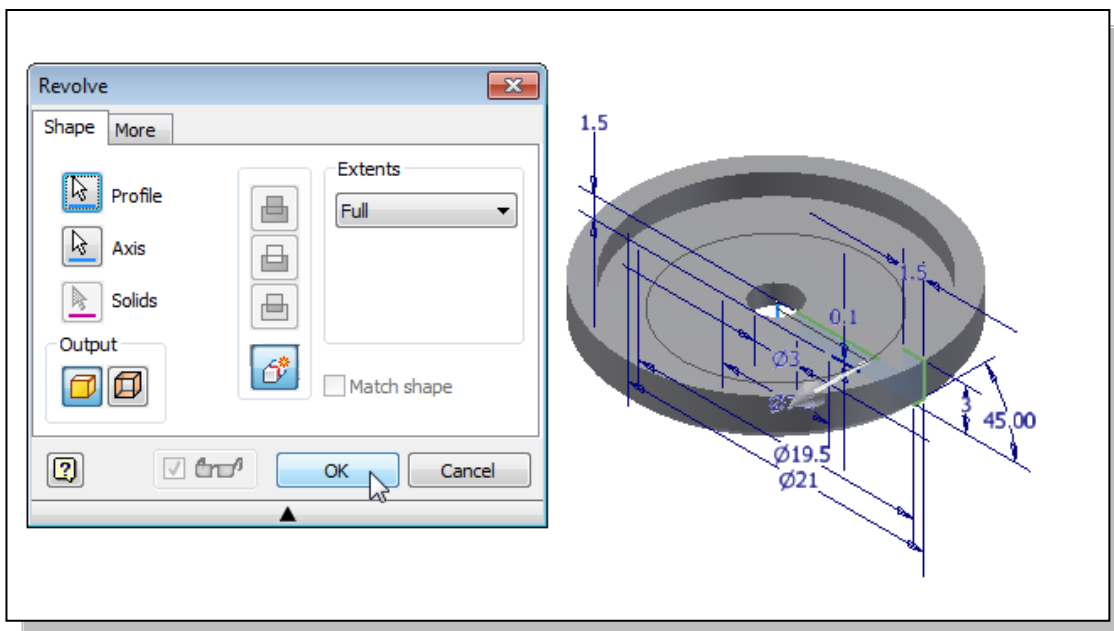
16. Click **OK** to accept the selection and start a new file.

17. In the *Quick Access* menu, select **Save** to save the model.

Save (Ctrl+S)
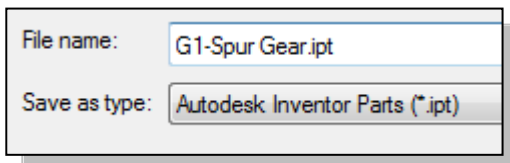
18. Enter the **G1-Spur Gear** as the part file name and save the model. Note that additional modifications will be performed in the next sections.

File name: G1-Spur Gear.ipt

Save as type: Autodesk Inventor Parts (*.ipt)

Export/Import the Generated Gear Profile

- We will create the *teeth* by importing the tooth profile from the gear generated by the *Design Accelerator*.
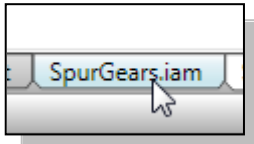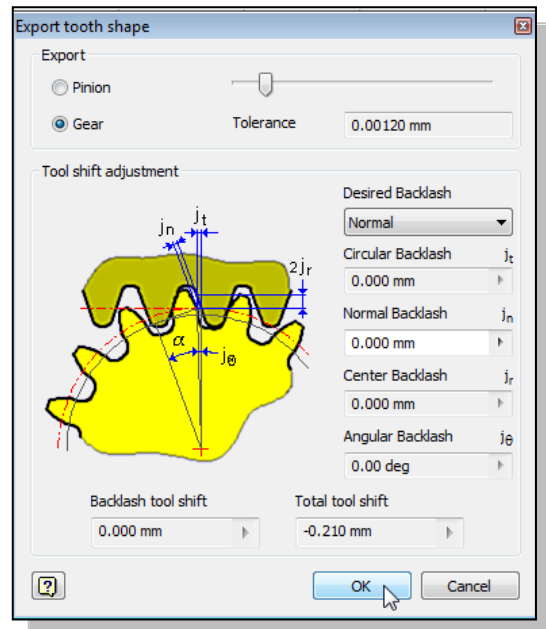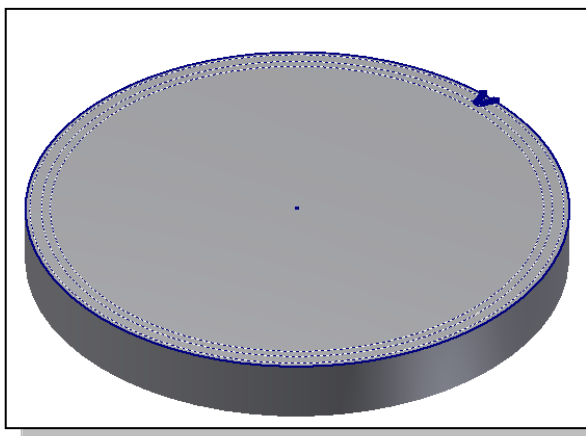
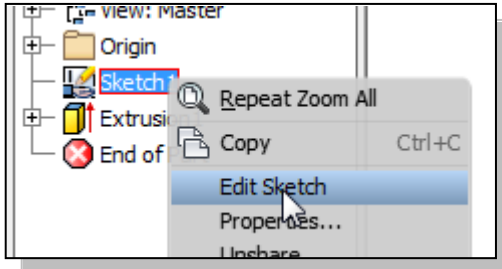1. Switch to the **SpurGears** assembly by clicking on the associated tab.
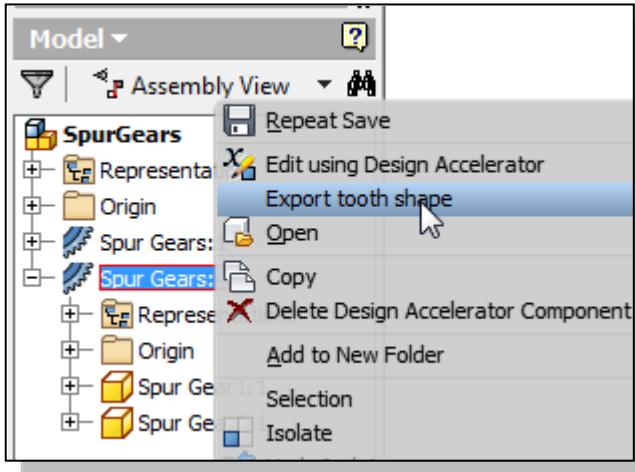
2. Inside the *browser* window, right-mouse-click on the second set of **SpurGears** to bring up the option menu as shown.

3. In the option menu, select **Export tooth shape** to proceed to export the tooth profile.

4. In the *Export tooth shape* dialog box, select the **Gear profile** to export.
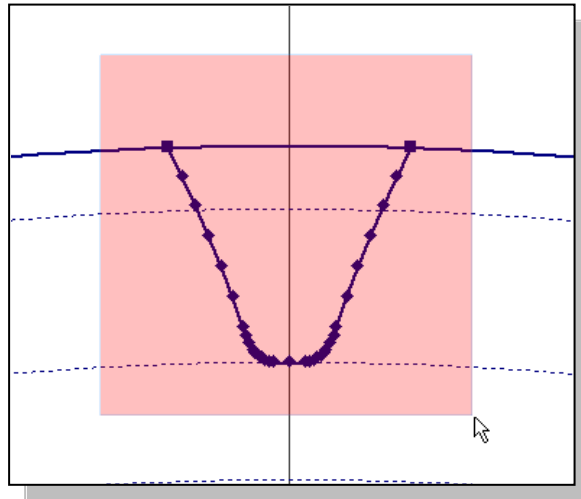
5. Click **OK** to exit the command.

- Note the tooth profile with of the selected gear is displayed in a separate window.
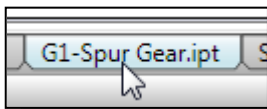
6. Inside the *browser* window, right-mouse-click on the **Sketch1** to bring up the *option menu* and select **Edit Sketch** as shown.
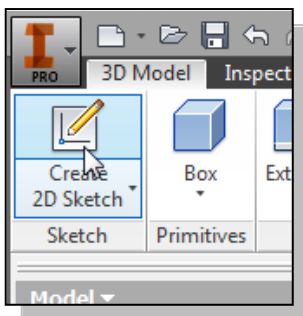
7. In the graphics window, use the left-mouse-button to enclose the tooth profile inside a **selection window** as shown.
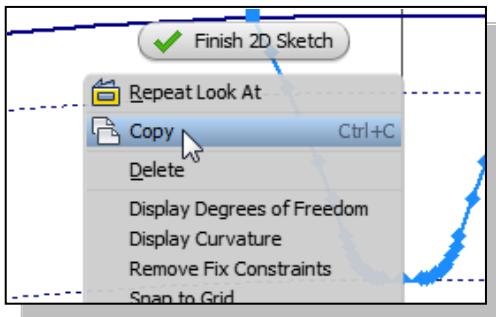
8. In the graphics window, right-mouse-click on bring up the option m*enu* and select **Copy** as shown.
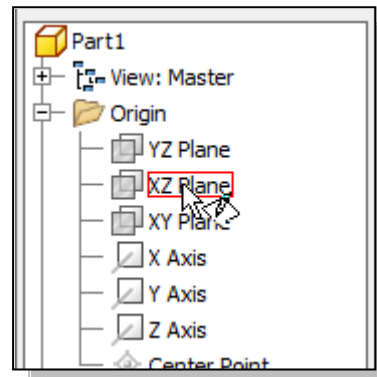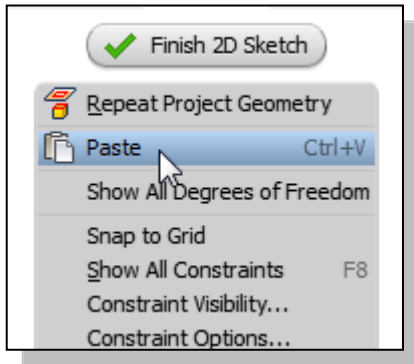
9. Switch to the **G1-Spur Gear** part by clicking on the associated tab.

10. In the *Sketch* toolbar, select the **Create 2D Sketch** command by left-clicking once on the icon.

11. Inside the *browser* window, select the XZ Plane as the sketching plane as shown.

Finish 2D Sketch

Repeat Project Geometry

Paste   Ctrl+V

Show All Degrees of Freedom

Snap to Grid

Show All Constraints   F8

Constraint Visibility…
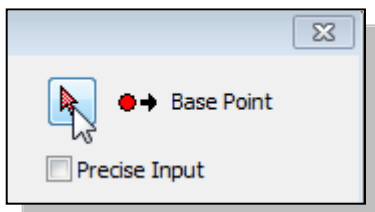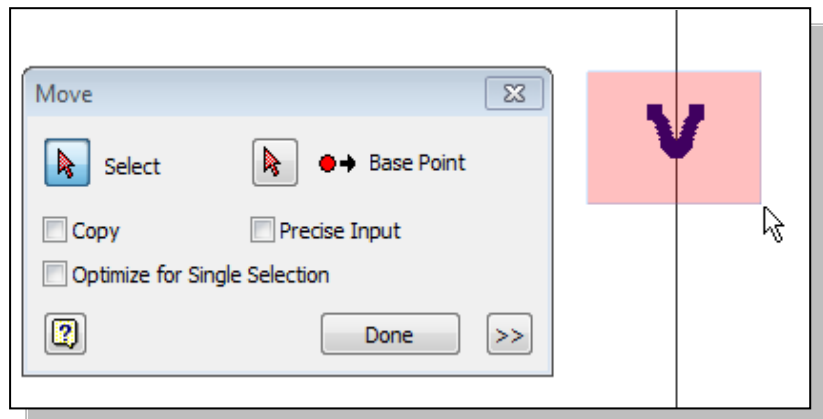
Constraint Options…

12. In the graphics window, right-mouse-click to bring up the option menu and select **Paste** as shown.

- Note the tooth profile is now imported into the current sketch. The imported geometry is placed above the solid model.

Move   **Trim**
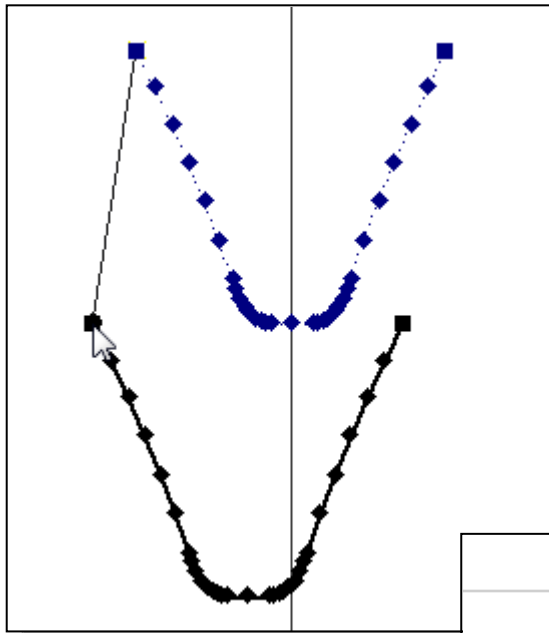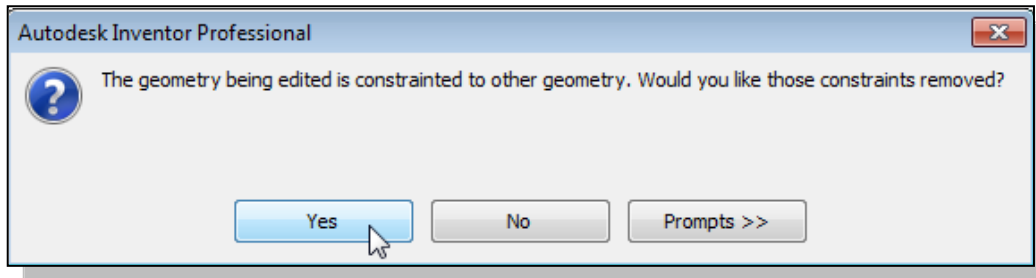
Move

Moves selected sk
specified, with an
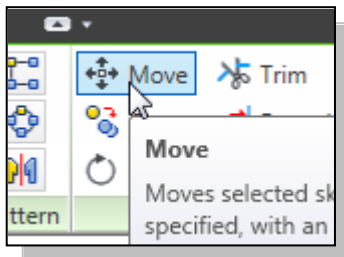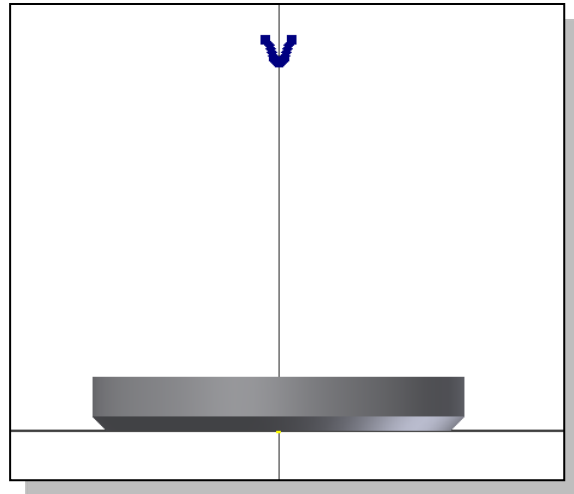
13. In the *Modify* toolbar, select the **Move** command by left-clicking once on the icon.

14. In the graphics window, use the left-mouse-button to enclose the tooth profile inside a **selection window** as shown.
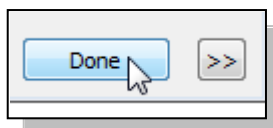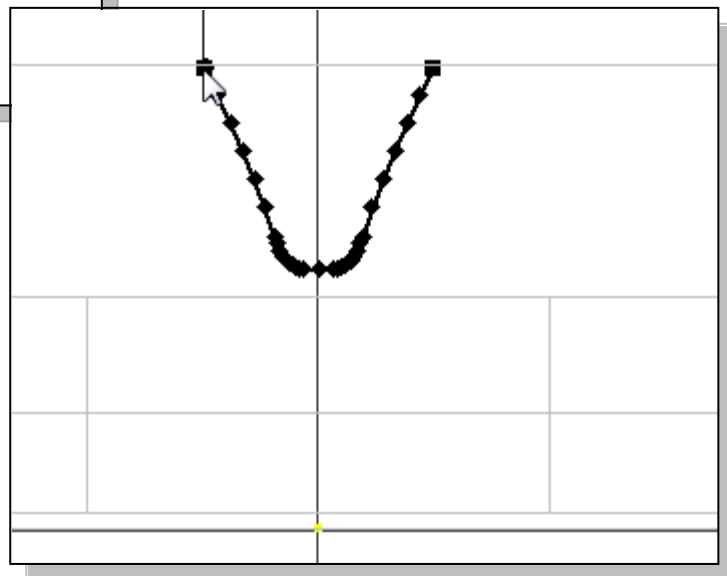
Move

Select   Base Point

☐ Copy   ☐ Precise Input

☐ Optimize for Single Selection

☐   Done   >>

Base Point

☐ Precise Input

15. In the *Move* dialog box, click on the **Base Point** icon as shown.

16. The message "*The geometry being edited is constrained to other geometry. Would you like those constraints removed?*" appears on the screen. Click **Yes** to allow adjustments to the selected geometry.

Autodesk Inventor Professional

The geometry being edited is constrainted to other geometry. Would you like those constraints removed?

Yes   No   Prompts >>

17. Select the **first node point** on the upper left hand side of the selected profile as the base point as shown.

18. Move the selected geometry directly downward and align the node to the **top** edge of the current solid model. Note the bottom node point is also aligned to the center axis.
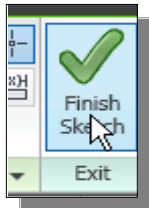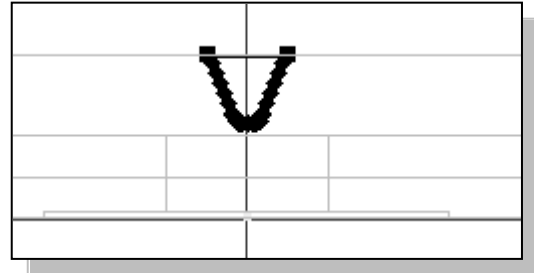
Done   >>

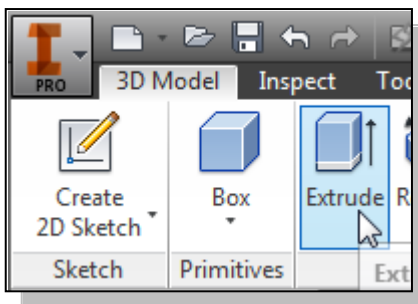19. Click **Done** to accept the change and exit the Move command.
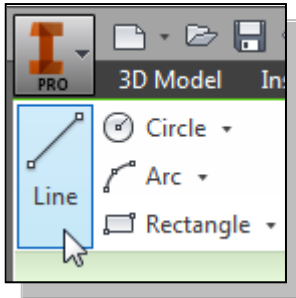
20. Select the **Line** command by clicking once with the left-mouse-button on the icon in the *Draw* toolbar.

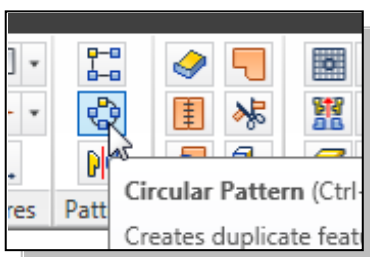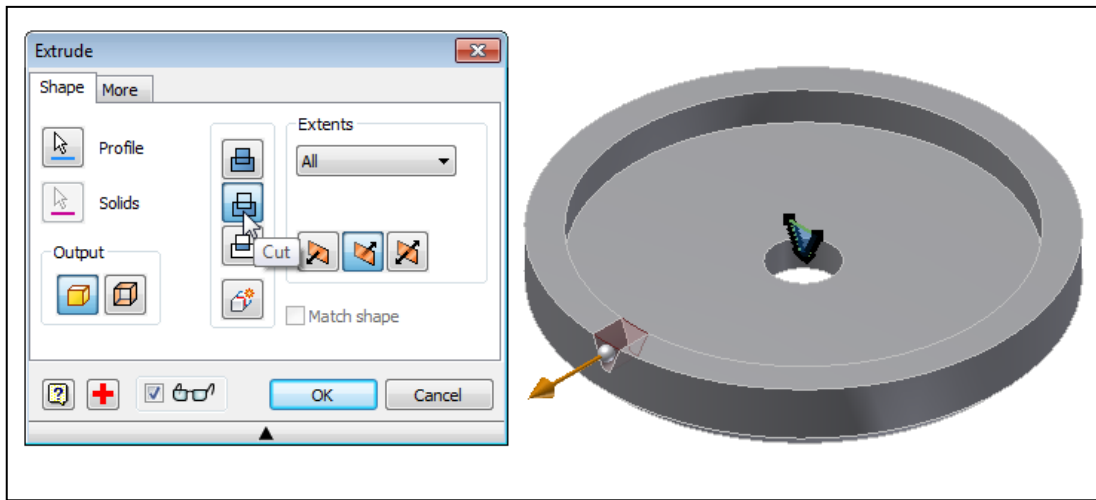21. Create a horizontal line connecting the two top node points as shown.

22. Select **Finish Sketch** in the *Ribbon* toolbar to exit the Sketch mode.
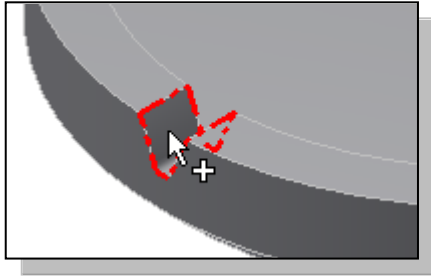
23. In the *Create* toolbar, select the **Extrude** command by clicking the left-mouse-button on the icon.

24. Select the inside region of the 2D sketch profile as shown.

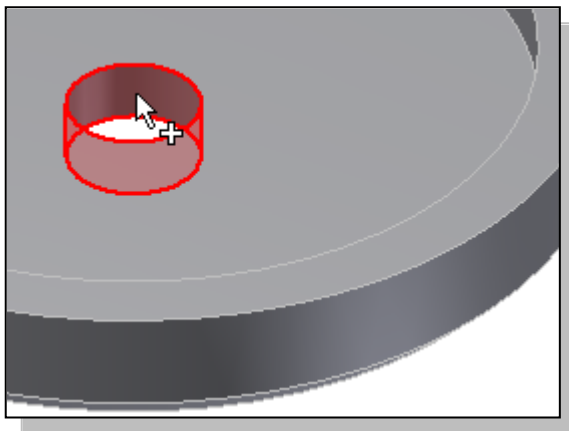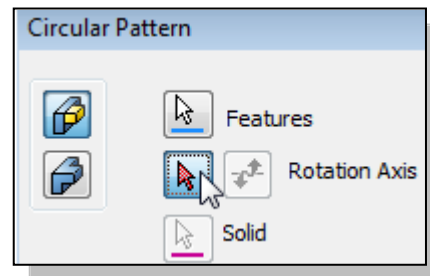25. On your own, complete the **Cut feature** as shown.

26. In the *Pattern* toolbar, select the **Circular Pattern** command by clicking the left-mouse-button on the icon.

27. In the graphics window, select the cut feature we just created.

28. In the *Circular Pattern* dialog box, click on the **Rotation Axis** icon as shown.

29. Select the cylindrical surface of the revolved feature; the associated center axis is therefore used as the rotation axis.

30. In the *Circular Pattern* dialog box, enter **40** for the number of pattern features and **360 degrees** in the *Placement* section as shown.
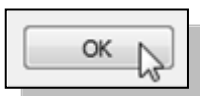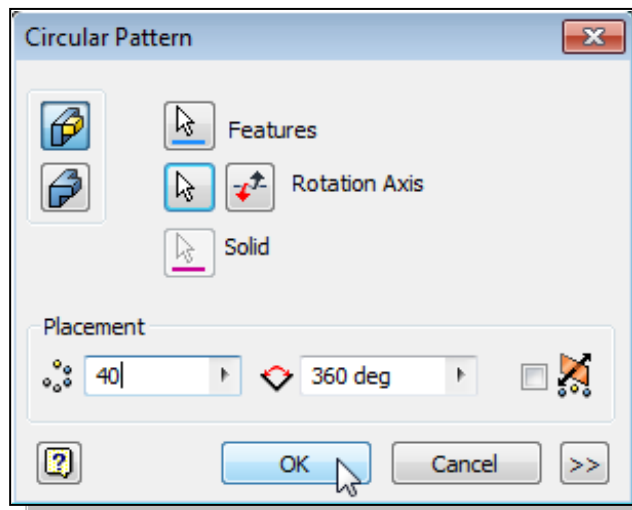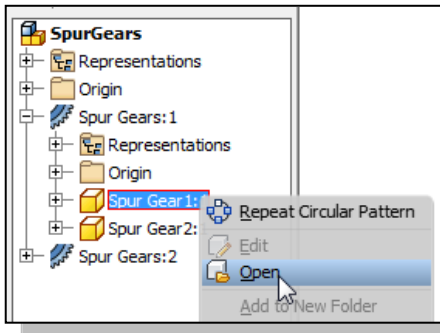

31. Click **OK** to accept the settings and create the circular pattern.

- The Circular Pattern command can be used to quickly create duplicates about a center axis of rotation.
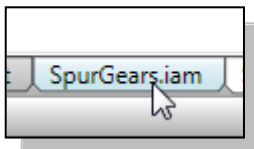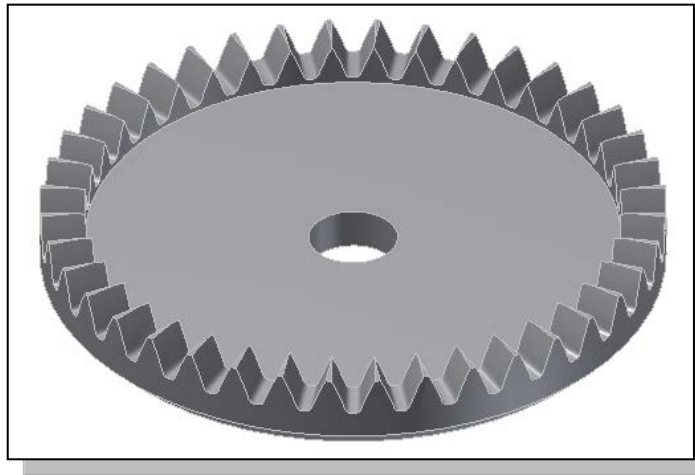
32. Switch to the **SpurGears** assembly by clicking on the associated tab.

33. Inside the *browser* window, right-mouse-click on the **Pinion Gear** of the first set of gears to bring up the option menu as shown.

34. In the option menu, select **Open** to open the selected model as shown in the figure.

35. On your own, copy and paste the profile of the *Pinion Gear* into a new 2D sketch on the top face of the **G1-Spur Gear** as shown.
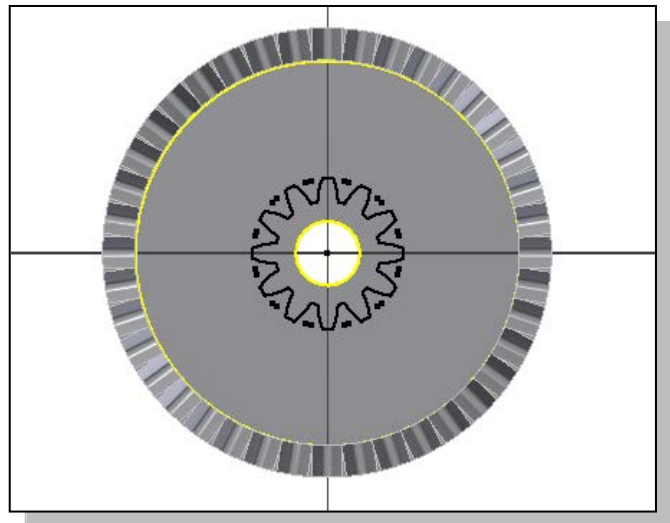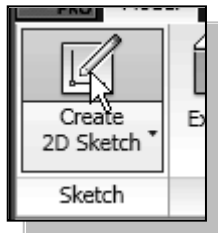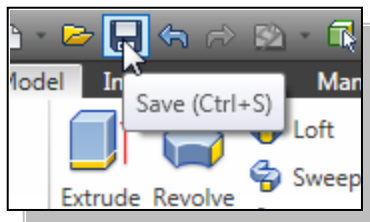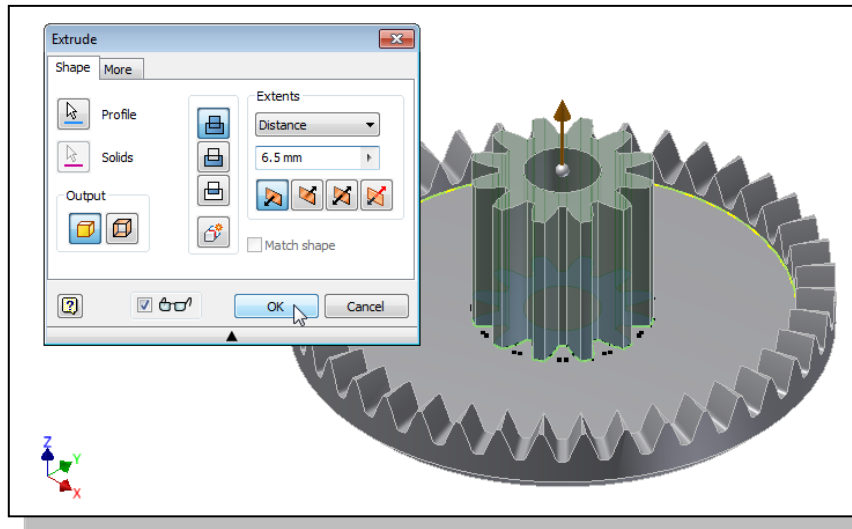
36. Select **Finish Sketch** in the *Ribbon* toolbar to exit the Sketch mode.
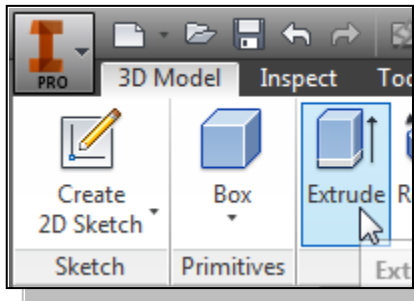
37. In the *Create* toolbar, select the **Extrude** command by releasing the left-mouse-button on the icon.

38. Select the inside region of the 2D sketch to create a profile as shown.

39. On your own, create the **6.5mm** extrusion feature as shown.

40. In the *Quick Access* menu, select **Save** to save the ***G1-Spur Gear*** model.

- All of the required gears for the *Tamiya Tiger* kit have been created.