Modeling of a Control lever shaft using Inventor 2014

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This tutorial consists of one lesson that will provide you with a basic understanding of modeling special parts, and introduce you to achieving advanced abilities in modeling parts.

Audience: Users accommodated with basic modeling in Inventor 2014

Prerequisites: Basic Autodesk Inventor 2014 knowledge

Time to complete: 20 minutes

Objectives

In this tutorial, you learn how to model a special part including the following:

• Creating multiple work planes for different sketches

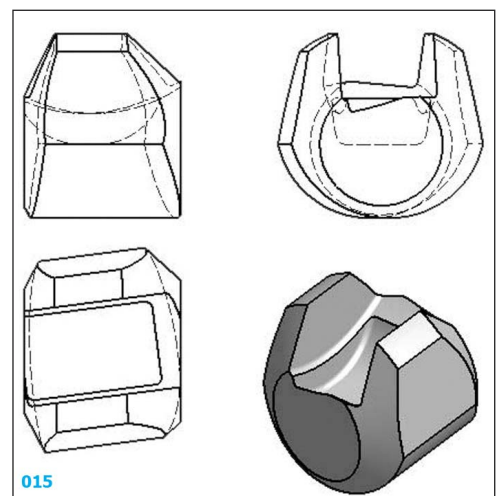• Obtaining the middle body of the part by using Intersection of volumes

Tutorial Files

No files necessary

Modeling the Control lever shaft

Part Analysis: The part is included in the steering sub assembly of some vehicles. It houses a roller which is engaged with the worm shaft connected to the steering wheel.
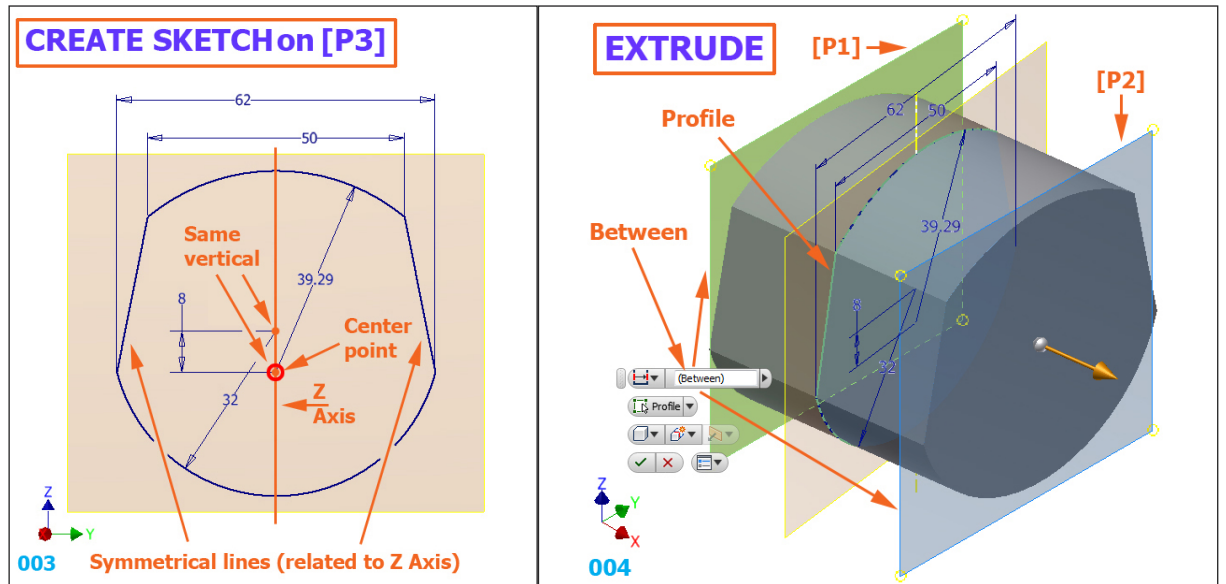
Multiple views of the middle body of part are presented in the image. The special construction consists of its spatial geometry.

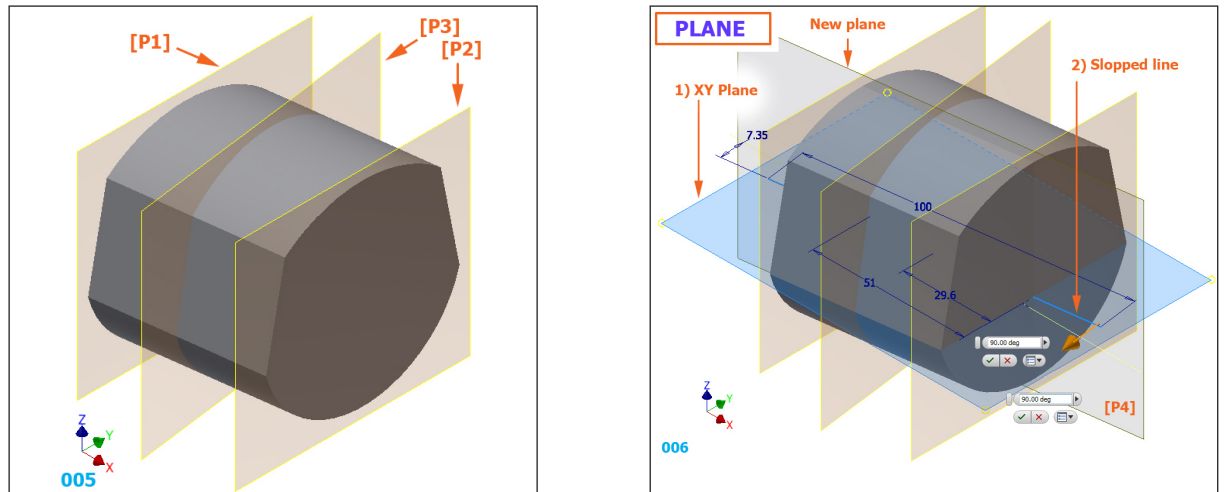The image with the entire part and its technical drawing follows.

015

Technical Drawing of the part

We have to abide all conditions from the technical drawing.

Procedure

The following 35 pictures illustrate the whole procedure. When a picture has a CAPITAL LABEL surrounded by a red border with a name of an Inventor tool, you have to prior activate that tool.

001. New sketch: a slopped line centered on *Center point* (length 100, slopped at 7 degrees 21' from the projected *X Axis;* two points (P1 and P2) at 29.6 and 51 are added on the slopped line

002. Three new work planes: [P1] and [P2] normal to *X Axis* through points P1 and P2, and [P3] normal to slopped line through *Center point;* select the line first and the point second

003. After taking visibility of [P1] and [P2] and the first sketch, a new sketch on [P3]
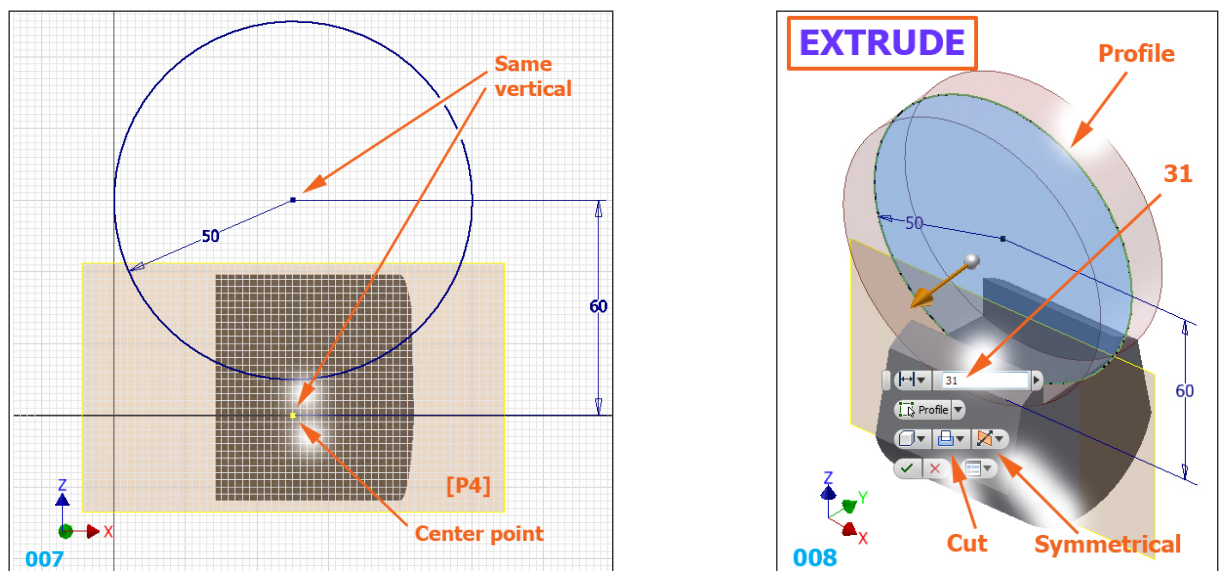004. **Extrude** the new profile between [P1] and [P2] (selected from *browser*)

CREATE SKETCH on [P3]

62

50

Same
vertical

39.29

8

Center
point

Z
Axis

32

Z

Y

003   Symmetrical lines (related to Z Axis)

EXTRUDE   [P1]→   [P2]

Profile

62 50

Between

39.29

32

(Between)

Profile

Z

Y

X

004

005. The result of extruding
006. New *work plane:* to create [P4] select successively 1 and 2, in order

[P1]   [P3]
[P2]

Z

Y

X

005

PLANE   New plane

1) XY Plane

2) Slopped line

7.35

100

51   29.6

90.00 deg

90.00 deg   [P4]

Z

Y

X

006

007. New sketch on [P4]: Circle with Radius=50 at 60 from *Center point* on the same vertical
008. **Extrude** the circle symmetrically on 31, option *Cut*

Same
vertical

50

60

[P4]

Z

X

Center point

007

EXTRUDE   Profile

31

50

31

Profile
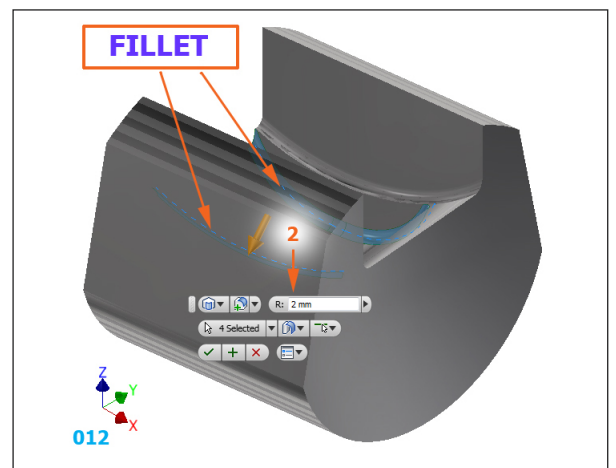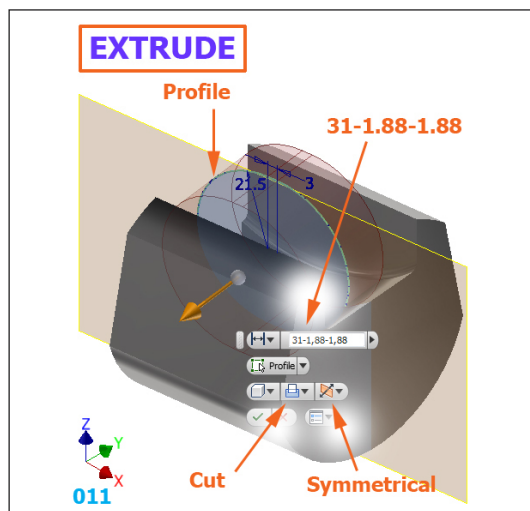
60

Z

Y

X

Cut   Symmetrical

008

009. **Fillet** Radius 2 on the indicated edges

010. New sketch on [P4]: two semicircles at 3 (the inferior tangent centered on *Center point*)

FILLET

R: 2 mm

2 Selected

2

009

Semicircle

21.5

3

Semicircle

Tangent line

Center point

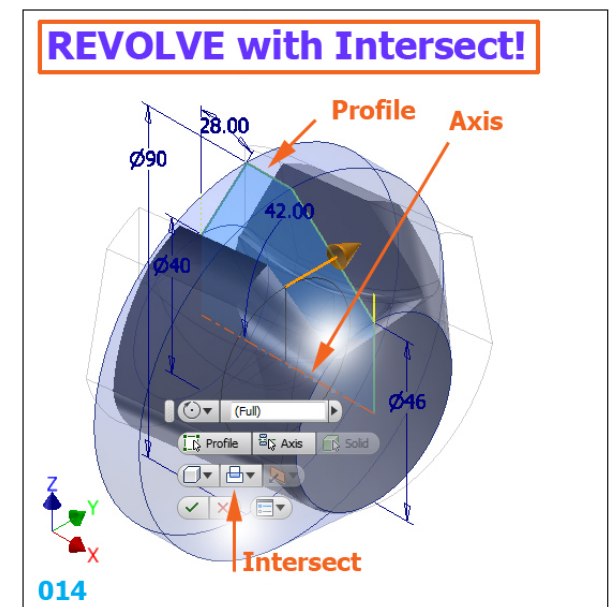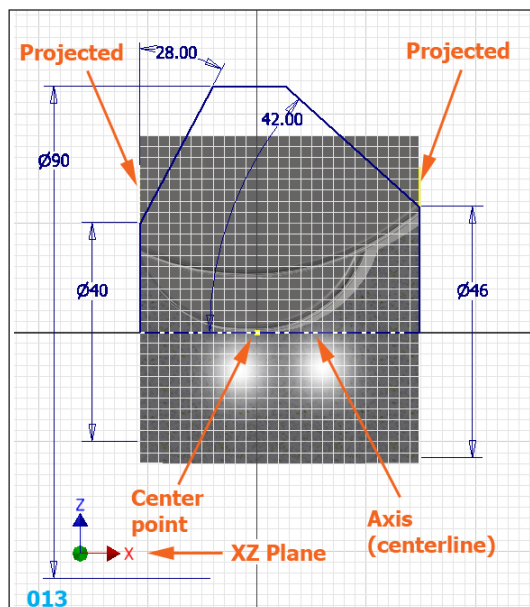Coincident (Midpoint)

[P4]

010
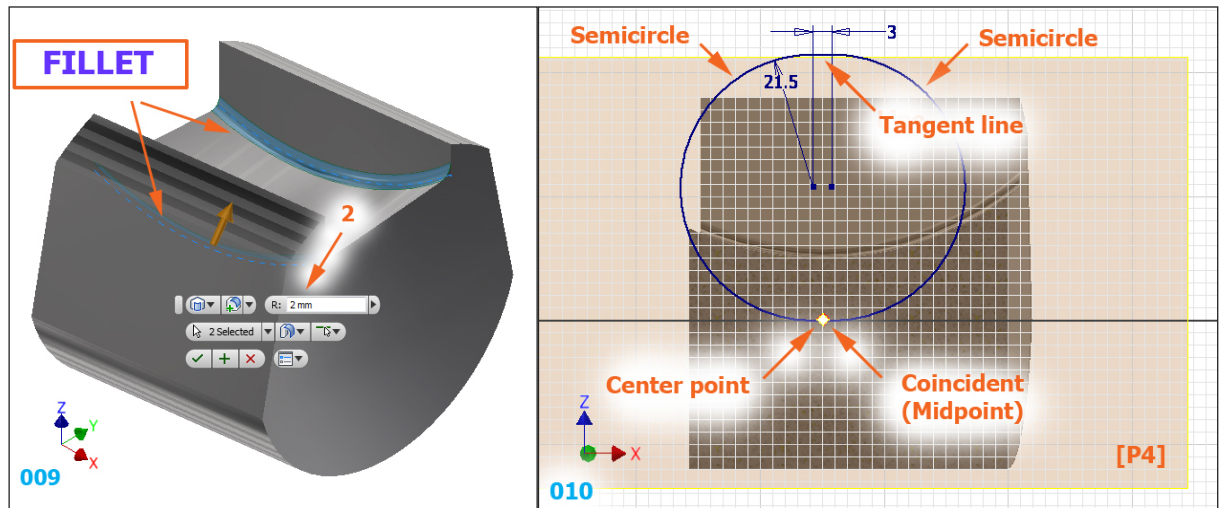
011. **Extrude** the closed profile symmetrically on 31-1.88-1.88 (as an expression)

012. **Fillet** Radius 2 on the indicated edges

EXTRUDE

Profile

31-1.88-1.88

21.5 3

31-1,88-1,88

Profile

Cut

Symmetrical

011

FILLET

R: 2 mm

4 Selected

2

012

013. New sketch on *XZ Plane:* the axis (centerline) is coincident with *Center point*

014. **Revolve** the closed profile using the *Intersect* option!

Projected

28.00

42.00

Projected

Ø90

Ø40

Ø46

Center point

XZ Plane

Axis (centerline)

013

REVOLVE with Intersect!

Profile Axis

28.00

Ø90

42.00

Ø40

(Full)

Profile Axis Solid

Ø46

Intersect

014

Modeling of a Control lever shaft using Inventor 2014

015. The middle body finished

016. **Revolve** the closed profile in a new sketch: the midpoints on the same vertical with *Center point*

015

REVOLVE

[P3]

65

50

Ø12

Ø18

23,39

Same vertical

Center point

016
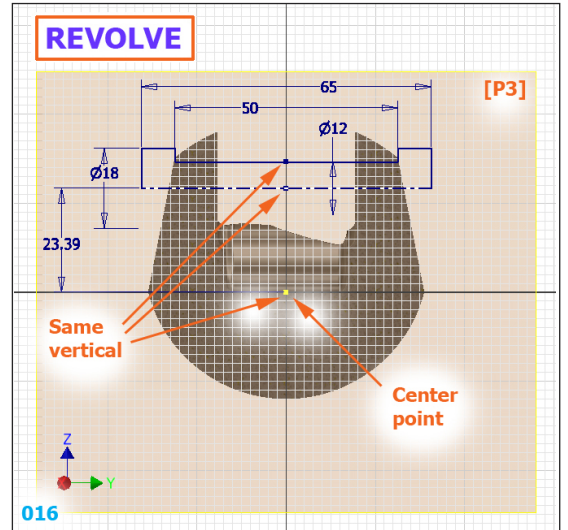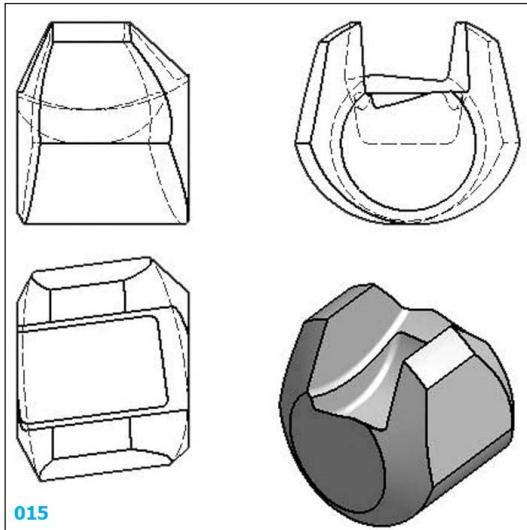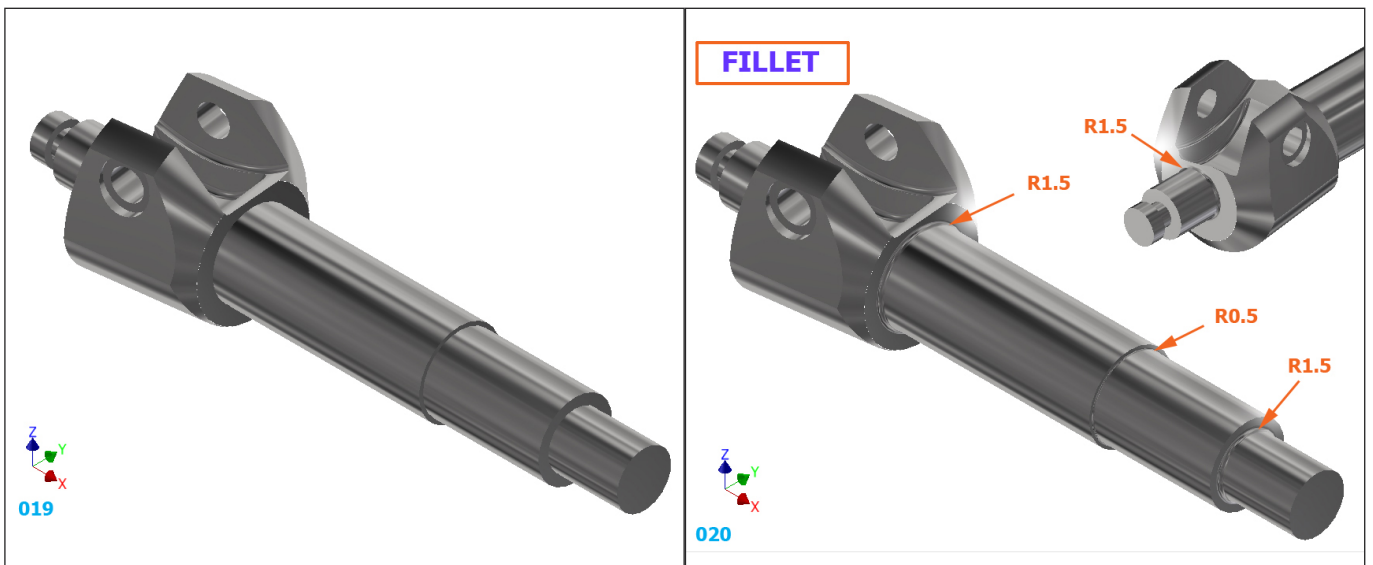
017. The result of revolving with *Cut* option

018. **Revolve** the ends of part with *Join* option using two closed profiles in a new sketch

017

REVOLVE

32

6

Ø16

6

Ø10.5

Ø21.1

Ø33

Ø30

Ø22

28

47

242

018
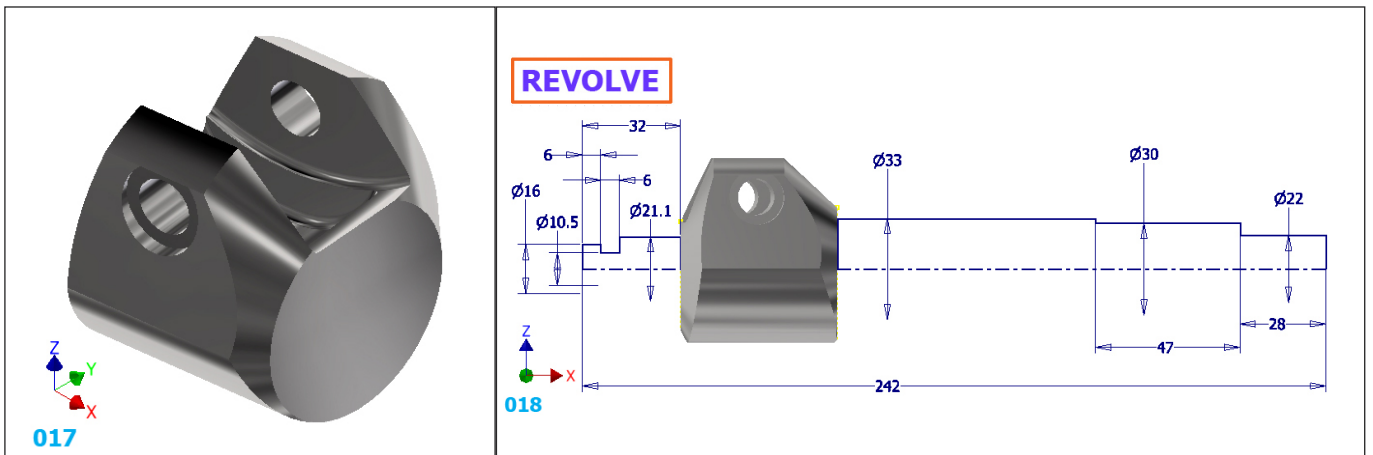
019. The result of revolving the two closed profiles

020. **Fillet** four edges with different radii

019

FILLET

R1.5

R1.5

R1.5

R0.5

R1.5

020

021. **Thread** the right end using the specification *ISO Metric profile / M22x3*
022. **Chamfer** five edges with different values

THREAD

Specification

Location

24.5

021

CHAMFER

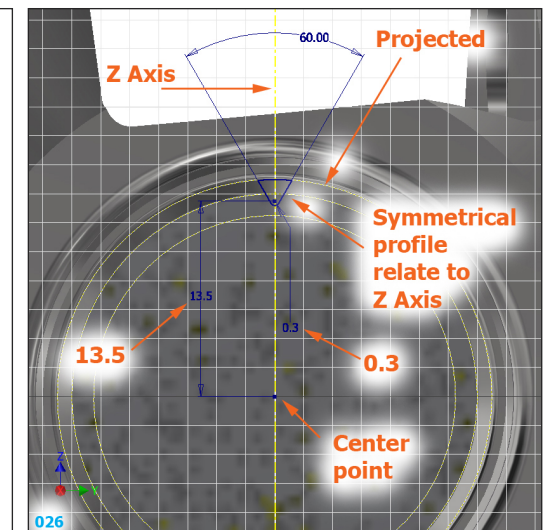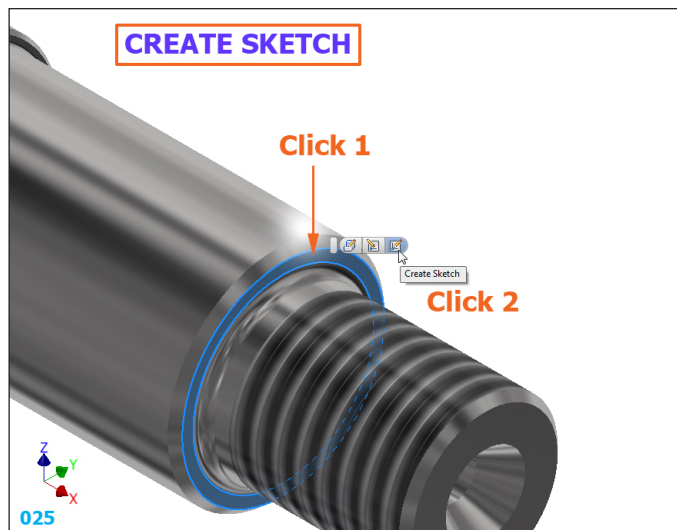0.5x45°
0.5x45°
0.5x45°
1.5x45°

022
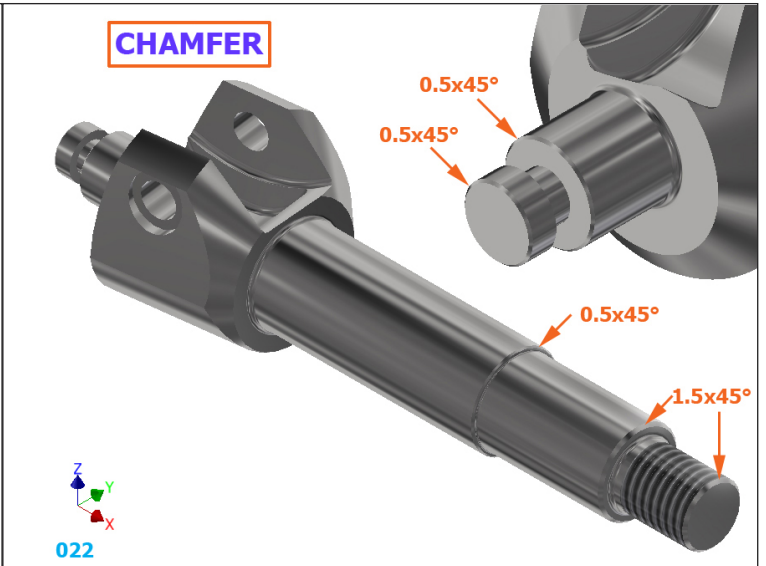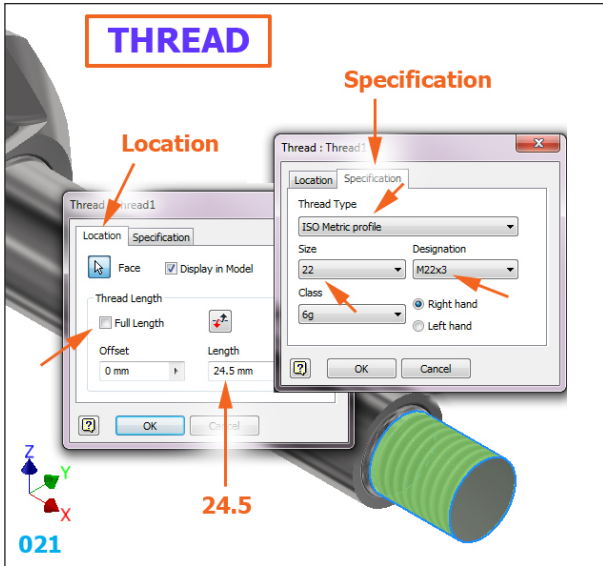
023. **Hole** on the right end of part using concentric option to obtain a countersunk hole at 60° (Ø3)
024. **Hole** on the left end of part using concentric option to obtain a countersunk hole at 60° (Ø6)


HOLE

023

HOLE

024
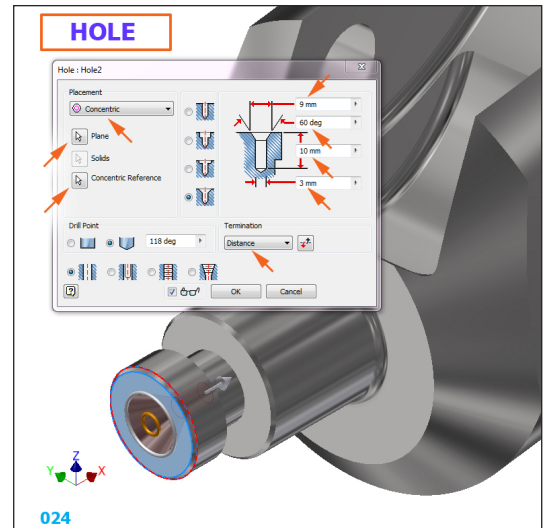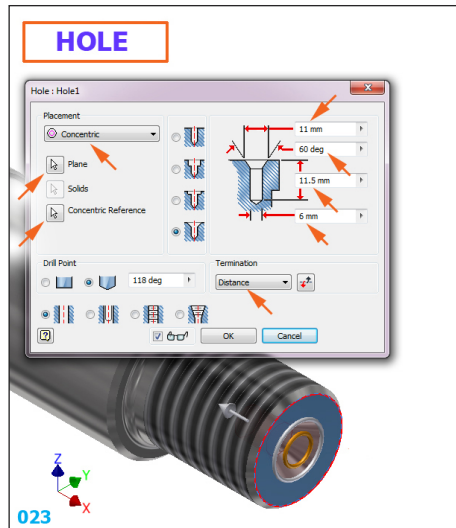
025. Preparing for creating a sketch in two steps: 1. Click on face, 2. Select „*Create Sketch*" icon
026. The sketch contains a pseudo triangle at 60° with a filleted peak of 0.3 at 13.5 from *Center point*

CREATE SKETCH

Click 1

Click 2

Create Sketch

025

60.00

Projected

Z Axis

Symmetrical profile relate to Z Axis

13.5

0.3

13.5

0.3

Center point

026

027. **Extrude** the „triangle" on a distance of 28, using opton *Cut*

028. Start creating a sketch in two steps: 1. Click on face, 2. Select „*Create Sketch*" icon

EXTRUDE

28

60.00

Projected circle

Cut

13.5 0.3

027

CREATE SKETCH

Click 2

Create Sketch

Click 1

028
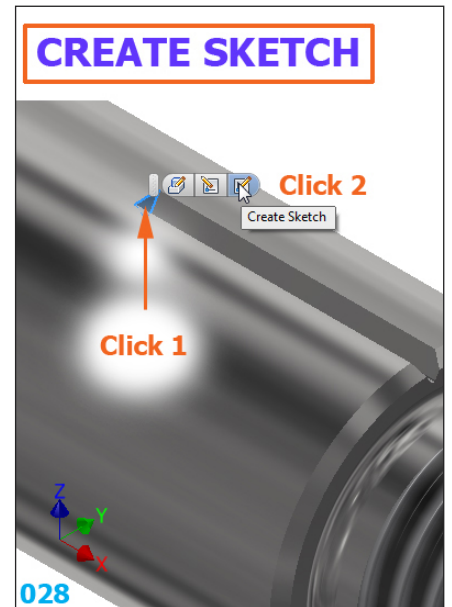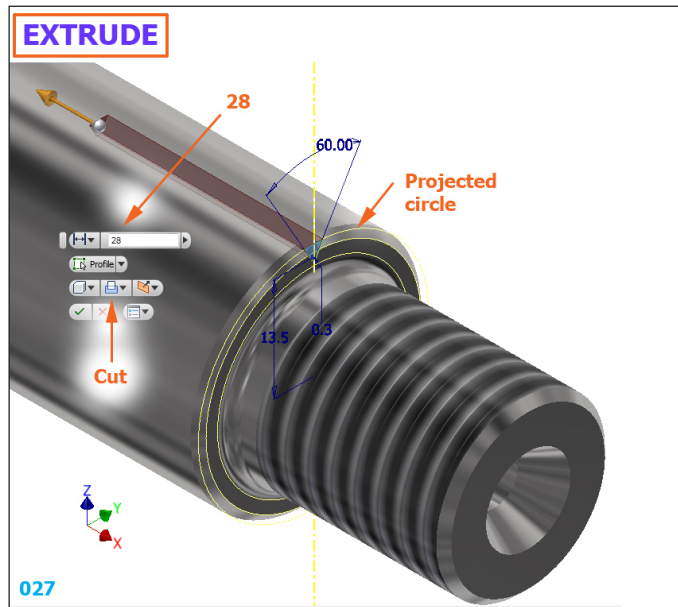
029. The sketch contains the projected „triangle" and a centerline (as Axis) at 53 from *Center point*

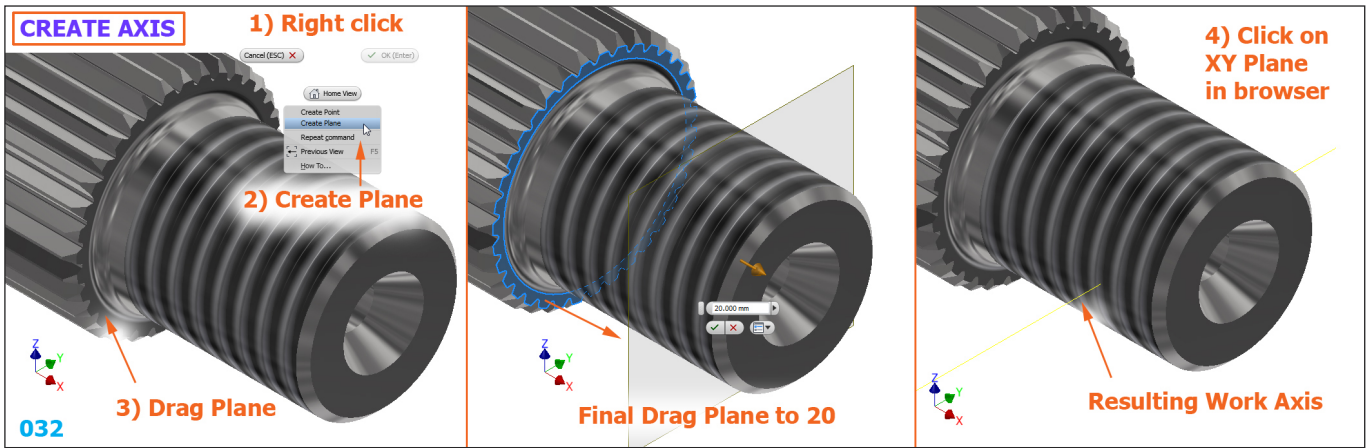030. **Revolve** the „triangle" around the Axis using the option *Cut*

Projected profile

53

Center point

029

REVOLVE

Axis

53

(Full)

Profile Axis Solid

Cut

030
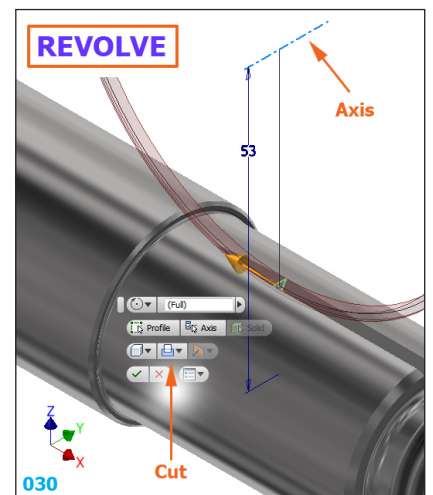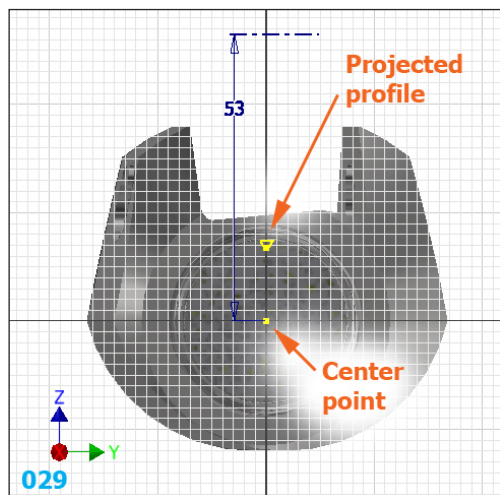
031. Apply Circular Pattern to last two features around the part axis for 35 times

CIRCULAR PATTERN

Circular Pattern

Features

Rotation Axis

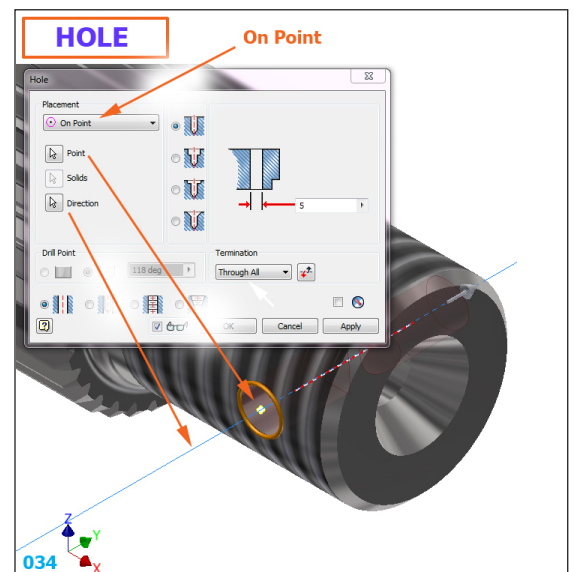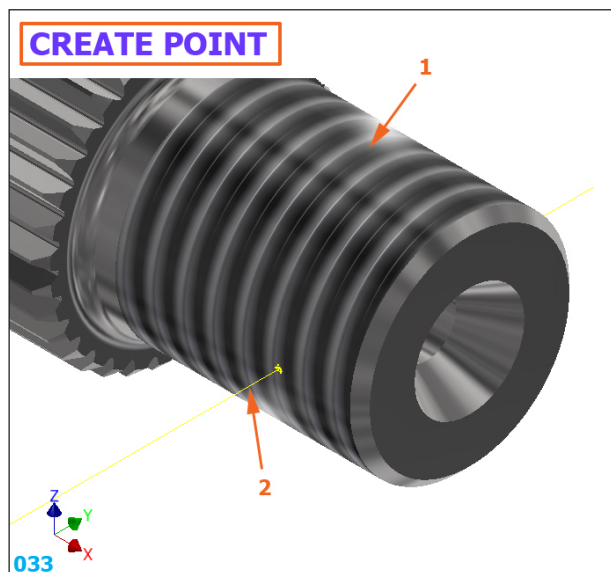Solid

Placement

35   360 deg

OK   Cancel   >>

35

031
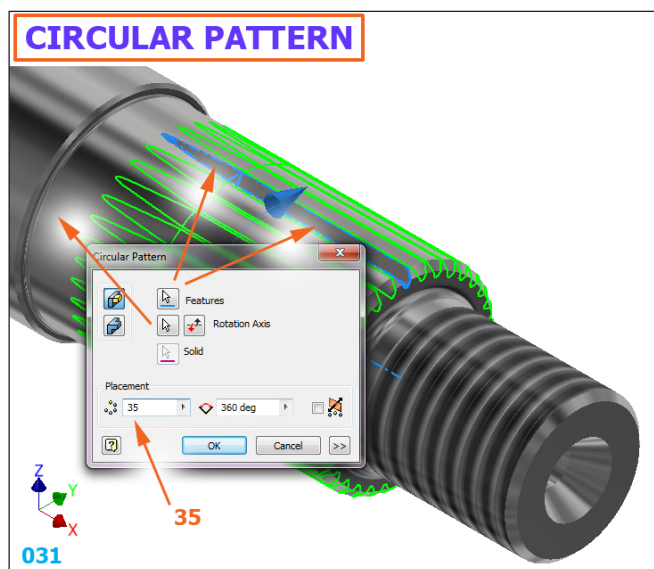
032. Create an *work axis* using a procedure in 4 steps, starting with right-click on empty graphic space

CREATE AXIS
1) Right click
2) Create Plane
3) Drag Plane
032

Final Drag Plane to 20

4) Click on XY Plane in browser
Resulting Work Axis

033. Create an *work point* by clicking successively on a face (1) and the axis (2)

034. Create **Hole** with the option *On point* by selecting the point and the axis, in this order

CREATE POINT
1
2
033

HOLE   On Point
034

035. Final part

035

End of lesson
☐