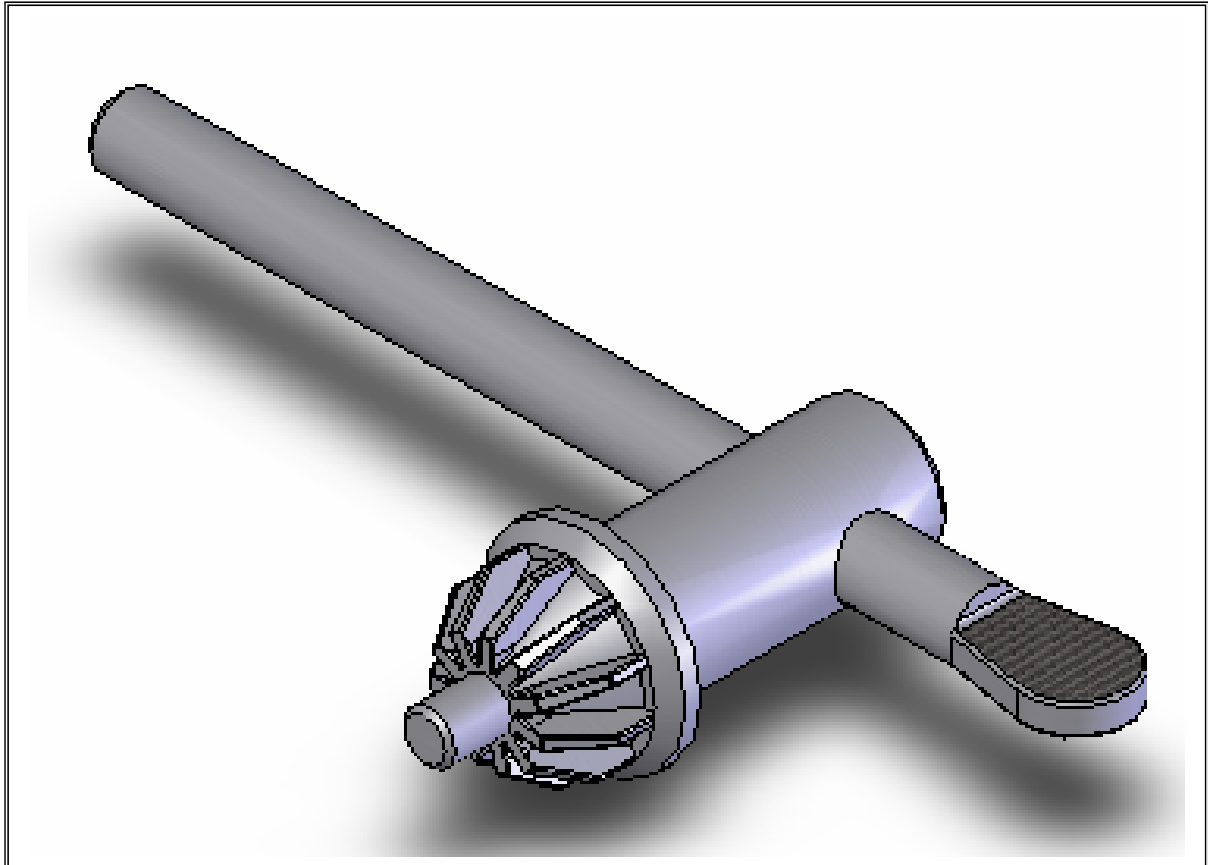




## Drill Chuck Key



### **Prerequisite Knowledge**

To complete this model you should have experience of the following:  
Sketching, Extrude Boss/Base.

### **Focus of Lesson**

Revolve Boss/Base, Revolve Cut, Create a Spline Sketch, Use “Circular Pattern” features.

### **Commands used:**

**Circle, Smart Dimension, Extrude Boss/Base, Linear Pattern, Shell, Fillet & Edit Material.**



## Cad TPN Module 3 Exercise No: 8 Drill Chuck Key

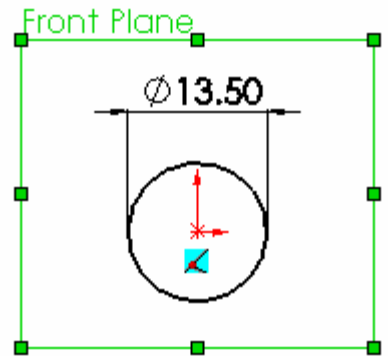


### Approach to Lesson:

#### Create sketch of Chuck Body

Create a circle on the **Front Plane** using the **Circle sketch** command and use **Smart Dimension** to dimension the sketch.

Rename the sketch as “**Body Sketch**”.



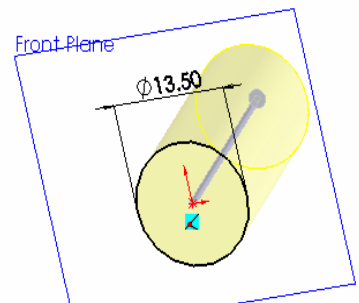
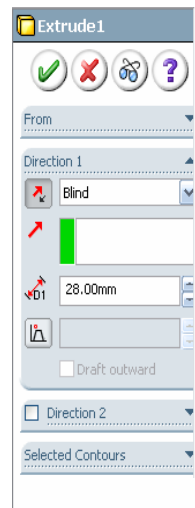
#### Extruded Boss/Base

Extrude the sketch using **Extruded Boss/Base** command.

Select **Blind** as the end condition and a distance of 28mm.

Apply a 1mm chamfer to “rear” body face.

Rename the feature as ‘**Body of Key**’.

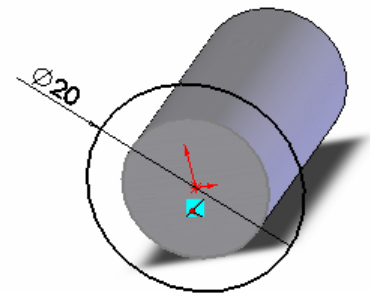


#### Create sketch for the Spline Base

Select the front face of the body as the plane for this sketch.

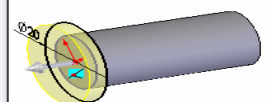
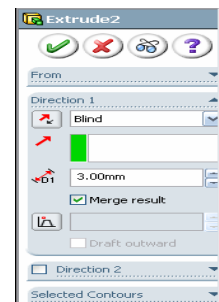
Sketch with **Circle** and **Smart Dimension**.

Rename sketch as “**Spline Base Sketch**”.

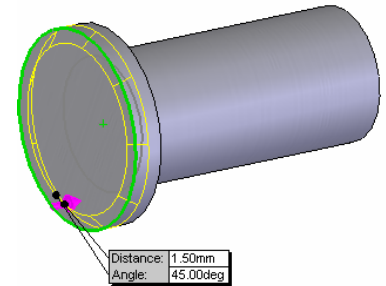


#### Extruded Boss/Base for Base of Spline

Extrude the sketch using **Blind** as the end condition and a distance of 3mm. Rename the feature as ‘**Spline Base**’.



**Apply Chamfer to the Base** Apply a 1.5mm chamfer to the outside edge of the base using the **Chamfer** feature.

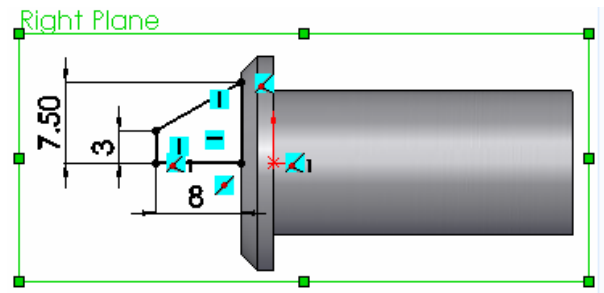


Rename the feature as “**Chamfer at base edge**”.

**Create a Spline Body Sketch**

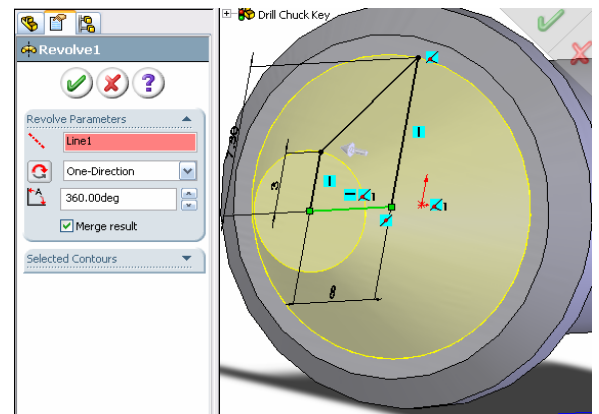
Using the **Line Sketch** command & **Smart Dimension**, draw the sketch on the Front Plane.

Rename the sketch as “**Spline Body Sketch**”.



**Revolved Boss/Base**

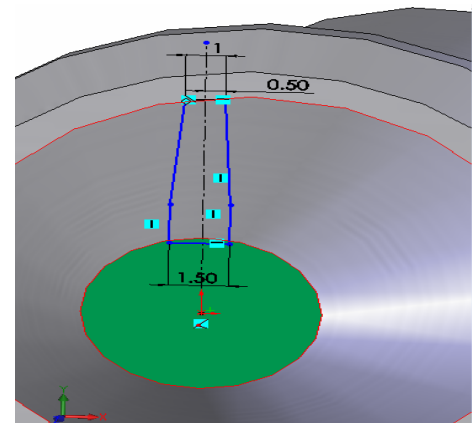
Using **Revolved Boss/Base**, revolve the sketch around the horizontal line in the sketch. Rename the feature as “**Revolved Spline Body**”.



**Create a Spline Sketch**

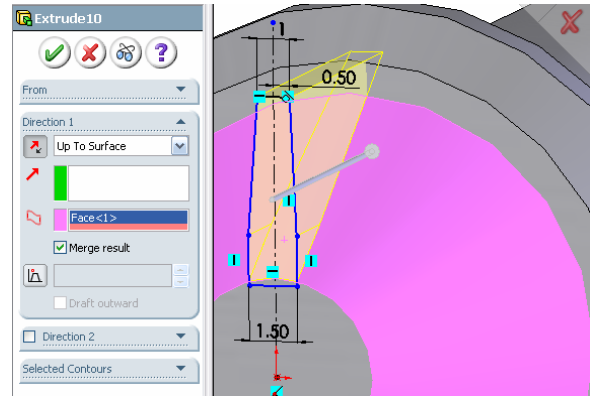
Select the End Surface of the “**Revolved Spline Body**”. Create a sketch using the **Line Sketch** & **Smart Dimension** commands.

Rename the sketch as “**Spline Sketch**”.



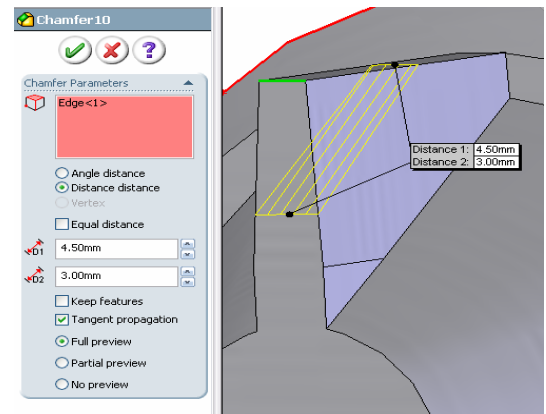
### Extruded Boss/Base

Use **Extruded Boss/Base** command to extrude this sketch up to the “surface” of the **Spline Body**. Rename the feature as “**Single Spline**”.



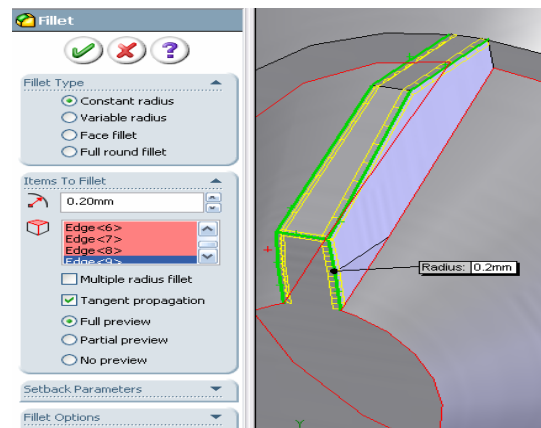
### Chamfer on Spline

Apply a “**Distance/Distance**” chamfer to the top edge of the spline of 4.5mm & 3mm. Rename the chamfer as “**Spline Chamfer**”.



### Fillet on Spline

Using the **Fillet Feature** command, apply a 0.2mm “**Constant Radius**” fillet to the Top & Front edges of the spline. Rename the feature as “**Spline Fillet**”.

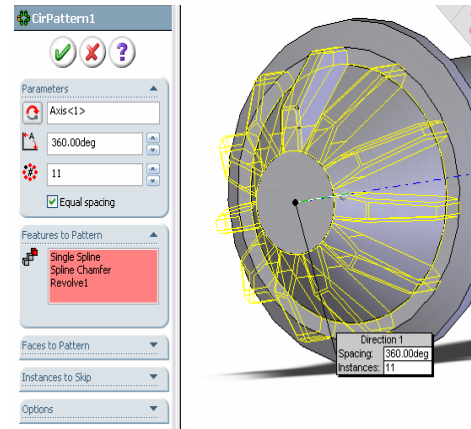




### Circular Pattern

Using “**Circular Pattern**” features command, pattern the Spline feature 11 times around a “Temporary Axes” obtained from the “View” drop down menu.

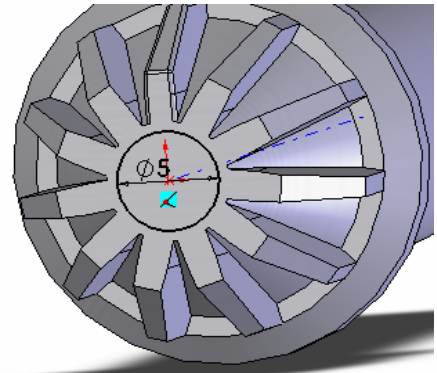
Rename feature as “**Spline Pattern**”.



### Pin Sketch

Sketch a diameter 5mm circle on the “front face” of the Body.

Rename the sketch “**Pin Sketch**”.



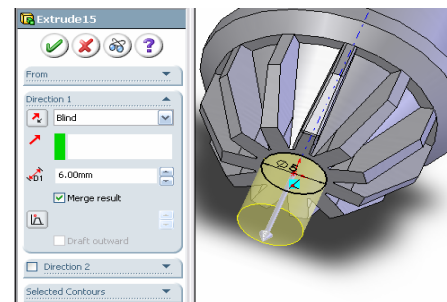
### Extruded Boss/Base

**Blind Extrude** the Pin Sketch a distance of 6mm.

Rename the feature as “**Pin Extrude**”.

Apply a 2mm chamfer to the top surface of the Pin Body

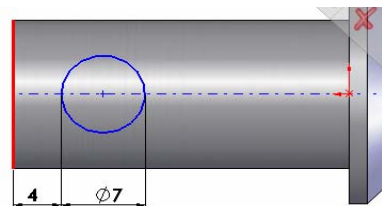
Rename the feature as “**Pin Chamfer**”.



### Handle Sketch

Sketch a **circle** on the Right Plane & **Smart Dimension** it.

Rename sketch as “**Handle Sketch**”.





### Cad TPN Module 3 Exercise No: 8 Drill Chuck Key

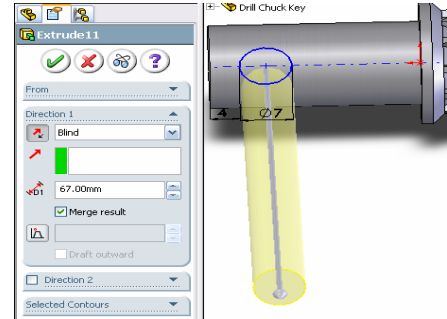


#### Extruded Boss/Base

**Blind Extrude** the handle sketch 67mm. Place a 1mm “chamfer” on the end of the handle.  
Rename feature as “**Handle Extrude 1**”.

Apply a 1mm **Chamfer** to the face of the handle.

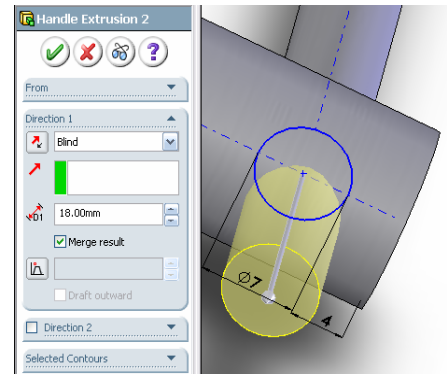
Rename the feature as “**Handle Chamfer**”.



#### Extruded Boss/Base

“Show” the Handle Sketch and extrude it by 18mm.

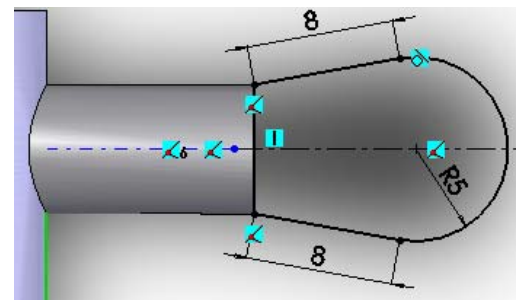
Rename the feature “**Handle Extrusion 2**”.



#### Thumb Grip Sketch

Using the **Line Sketch & Smart Dimension** commands sketch the thumb grip.

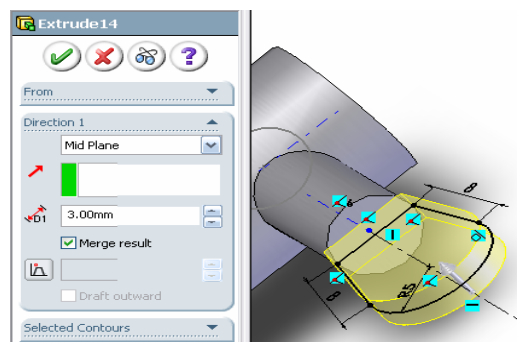
Rename the sketch as “**Thumb Grip Sketch**”.



#### Extruded Boss/Base

**Mid Plane Extrude** the Thumb Sketch 3mm.

Rename feature as “**Extruded Thumb Grip**”.

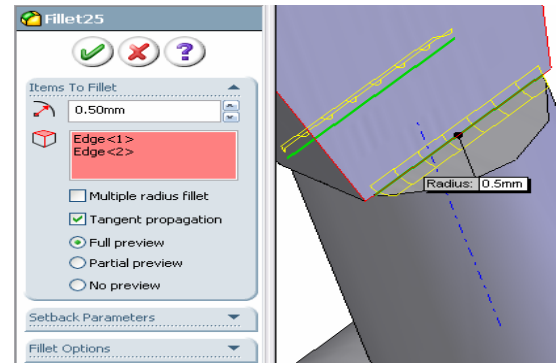




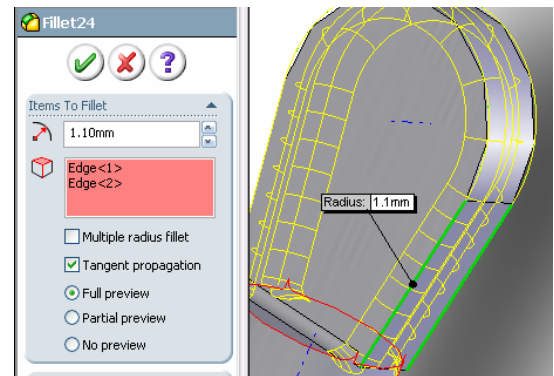
## Fillets

Apply a 0.5mm fillet using a  
“Constant Radius Fillet”.

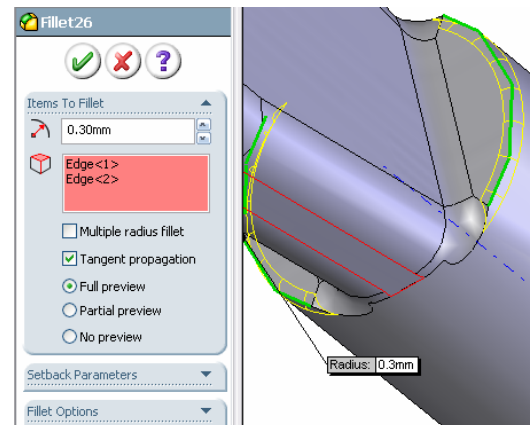
Rename feature “**Thumb Fillet 1**”.



Apply a 1.1mm “Constant Round”  
fillet & rename fillet as “**Thumb  
Fillet 2**”.



Apply a 0.2mm “Constant Round”  
fillet to the outer edge of the handle &  
rename the feature “**Thumb Fillet 3**”.



## Edit Material

“Cast Carbon Steel”.

Apply a “**Knurl 2**” type texture/appearance  
to both “**Thumb Grip**” surfaces.

### Cad TPN Module 3 Exercise No: 8 Drill Chuck Key

The technical drawing includes the following views and dimensions:

- Front View (Left):** Shows a central shaft with a diameter of  $\phi 20$  and a length of  $1.50$ . The shaft has chamfered ends with radii of  $R0.25$  and  $R0.35$ . A diameter of  $\phi 20$  is also indicated for the main shaft section.
- Side View (Top):** Shows a handle with a diameter of  $\phi 28$  and a length of  $6.1$ . The handle has a chamfered end with a radius of  $R1.75$  and a diameter of  $\phi 28$ . The handle is attached to the shaft with a diameter of  $\phi 1$  and a length of  $1$ . The handle has a chamfered end with a radius of  $R0.5$  and a diameter of  $\phi 1$ . The handle has a chamfered end with a radius of  $R0.5$  and a diameter of  $\phi 1$ .
- Top View (Bottom):** Shows a diameter of  $\phi 1.50$  for the handle section.
- 3D Model (Right):** A perspective view of the drill chuck key, showing the handle, the shaft, and the chuck mechanism.

<b>CAD-TPN MODULE 3</b>		DRAWN BY: YOUR NAME HERE		DATE: 09/12/2020	SCALE: 1:1	DATE: 09/12/2020	SHEET: 1 OF 1
<b>Drill Chuck Key</b>		DRAWN BY: YOUR NAME HERE		DATE: 09/12/2020	SCALE: 1:1	DATE: 09/12/2020	SHEET: 1 OF 1