

## **Introduction to Sweep - Allen Key part (A)**



Prerequisite Knowledge	e Previous knowledge of the following commands is required to complete this lesson, <b>sketching (line construction, dimensioning, polygon</b> ).
Focus of the Lesson	Sweep creates a base, cut, or surface by moving a <b>profile</b> (section) along a <b>path</b> .
	<ul> <li>The following conditions must be followed:</li> <li>The profile must be closed for a base or boss sweep feature; the profile may be open or closed for a surface sweep feature.</li> <li>The path may be open or closed.</li> <li>The path may be a set of sketched curves contained in one sketch, a curve, or a set of model edges.</li> <li>The start point of the path must lie on the plane of the profile.</li> <li>Neither the section, the path, nor the resulting solid can be self-intersecting.</li> </ul>
	<ul> <li>On completion of this lesson you will have the used the following aspects:</li> <li>Swept a sketch profile along a path to create swept boss / base.</li> <li>Examined the various sweep features.</li> </ul>
Commands Used	This lesson includes <b>Sketching (line, Smart Dimension, Polygon)</b> , <i>Swept Boss / Base</i> and <b>Edit Materials.</b>



Where to start? To create the model you first must identify the **path** and the **profile** required. Path Profile – Hexagon ront Plane



On which plane will the **sketch** for the path be constructed?







Note: There are two separate sketches required on separate planes to create the swept boss/base.

#### Selecting a sketch

Which sketch should be created first?

profile be constructed?

Either sketch can be constructed once both are connected by a common point.





Note: Both sketches will be constructed about the origin in this exercise, as shown.

#### TECHNOLOGY SUBJECTS SUPPORT SERVICE

## Sketch the Path on the front plane

Create the sketch using the **line** command and **smart dimension** as shown.

**Note:** The sketch is created from the origin (bottom left)





**Create Sketch** 

Create **polygon** sketch as shown from origin.







Note: Use inference line to ensure the hexagon is vertical.

Add relation

In order to make the hexagon vertical. Add relation to shown line and add vertical relation.



Dimension

Smart dimension sketch as shown.



Exit sketch and Rename

# Exit sketch and rename as **Hexagonal profile**.





#### Creating the feature

Select **Features** from the **Command Manager**. The **Features** toolbar has now replaced the **Sketch** toolbar along the top of the screen



Choose **Swept Boss/Base**, Swept Boss/Base the sketch rotates to a trimetric view with a preview of the proposed revolve.

OR



From the top toolbar select **Insert**, **Boss/Base** and **Sweep**.

TECHNOLOGY SUBJECTS SUPPORT SERVICE

Sweep Feature Settings

The **PropertyManager** appears as shown.

S 🛯 😫 🔶	
🚭 Sweep ?	
✓ ×	
Profile and Path	]
≪	
≪ ■	
Options 🛛 🕹	]

**Note:** The sweep feature cannot recognize which of sketches is the path and profile. Therefore all the selections are currently empty.

#### Lets analyse the Propertymanager

**Select Path and Profile** 

**Profile selection** 

**Profile** - sets the sketch profile (section) used to create the sweep.

Select the hexagonal profile



#### Path selection

SolidWorks will now look for a path.



Path

Ś



- sets the path along which the profile sweeps.

Options

You are offered two options to select

from.

1. Orientation/twist type

Controls the orientation of the **Profile** it sweeps along the **Path**.

Selection options:

- Follow Path. Section remains at same angle with respect to path at all times.
- **Keep normal constant**. Section remains parallel to the beginning section at all times



- **Note:** This will be looked at in part (b) of this exercise.
  - Follow path and 1st guide curve
  - Follow 1st and 2nd guide curves
  - **Twist Along Path**. Twists the section along the path. Define the twist by degrees, radians, or turns under **Define by**.
  - **Twist Along Path With Normal Constant**. Twists the section along the path, keeping the section parallel to the beginning section as it twists along the path

#### 2. Path alignment type:

Stabilizes the profile when small and uneven curvature fluctuations along the path cause the profile to misalign. (available with **Follow Path** selected above)



**Note:** Each of these selections is required for specific modeling exercises. This selection is not required for this exercise.

**Guide curves** 

Guides the profile as it sweeps along the path. Select guide curves in the graphics area.

This selection is not required in this exercise.

Guide <u>C</u> urves	~
5	1
Merge smooth faces	
ଟେ 1	3

FCHNOLOGY

-

SUBJECTS SUPPORT

SERVICE

Profile and Path

S Path

ons

Follow Path Path alignment type

None

Orientation/twist type

Merge tangent face:
 Show preview

as

#### Start/end tangency

This option allows you to add tangency constraints to the feature.

Start/End <u>T</u> angency	*
Start tangency type:	
None	•
End tangency type:	
None	•



#### Thin feature

Select thin feature by ticking the box in the top left corner as shown.

A window opens with a number of Options. Solidworks has entered some default parameters which can be adjusted.

The preview of the allen key changes to a thin wall sweep as shown.





Deselect the thin feature.

#### **Confirm Feature**

Click **OK** button *l***o** create the feature.

<b>S P P P</b>	
🗲 Sweep	?
<ul><li>✓ ×</li></ul>	
Profile and Path	~
🎸 🛛 Hexagonal Profile	
V Path	
Options	≽
Guide <u>C</u> urves	≽
Start/End <u>T</u> angency	≫
Thin Feature	≫



#### **Edit Materials**

Right hand click on the part name, select Edit Material.

#### Select Carbon steel from materials list.





Design & Communication Graphics



## Lesson Complete!





## **Other Possible Sweep Exercises**





Paper Clip

**Elbow Joint for waste pipe** 



Kitchen door handle



**Cooker Shelf** 



Mug









Design & Communication Graphics











