**Sweep**
Creates a solid with uniform cross section

**Loft (sweep and blend combined)**
Creates a solid with different cross sections, multiple paths could be used
Sweep Command

Creates a swept solid

Removes material as it sweeps
Sweep

Sweep command creates a solid by sweeping a profile along a specified path.

A sweep feature requires two sketches; a profile and a path. These two sketches cannot lie on the same plane or be parallel.

The path can be either open or closed profile. It could lie on one plane or lie on multiple planes (3D sketch)
Sweep Command in SW

Sweep using a profile, path and **Follow Path** option
Sweep Command in SW
Sweeps using different Options

- Follow Path
- Twisted
- Keep normal constant

Orientation/twist type:
- Follow Path
- Keep normal constant
- Follow path and 1st guide curve
- Follow 1st and 2nd guide curves
- Twist Along Path
- Twist Along Path With Normal Constant

Profile and Path:
- Sketch1
- Sketch3

Guide Curves:
Sweep Command in SW

Sweep using a profile, path and one guide
Sweep Command in SW
Sweep using a profile, path and **two** guides
Sweep Command in SW

Sweep using Merge Smooth Faces option
Sweep Command in SW
**Loft Command in SW**

*Loft* command creates a surface or a solid by sweeping and blending using two or more different profiles.
*Loft Command in SW*

**Start** and **End** Constraints options

- **Start constraint:**
  - Normal To Profile
  - None
  - Direction Vector
  - Normal To Profile

- **End constraint:**
  - Normal To Profile

Profile normal to the plane at the start

Profile normal to the plane at the end
Loft Command in SW

Loft using a single Guide Curve.
Loft Command in SW

Loft using many Guide Curves.

Guide 1
Guide 2
Guide 3
Guide 4

Guides 1,2
Guides 1,2,3
Guides 1,2,3,4
Loft Command in SW

Merging a Loft with a feature
10 spline sketches on planes .3 inches apart
Example
Knife handle
Knife blade

Hand and wrist