

# SOLIDWORKS GUIDE



## QUESTIONS

How remove dimension line (for a chain of dims in restricted area), but keep arrow?

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**\*\*\*\*\* INTRO \*\*\*\*\***

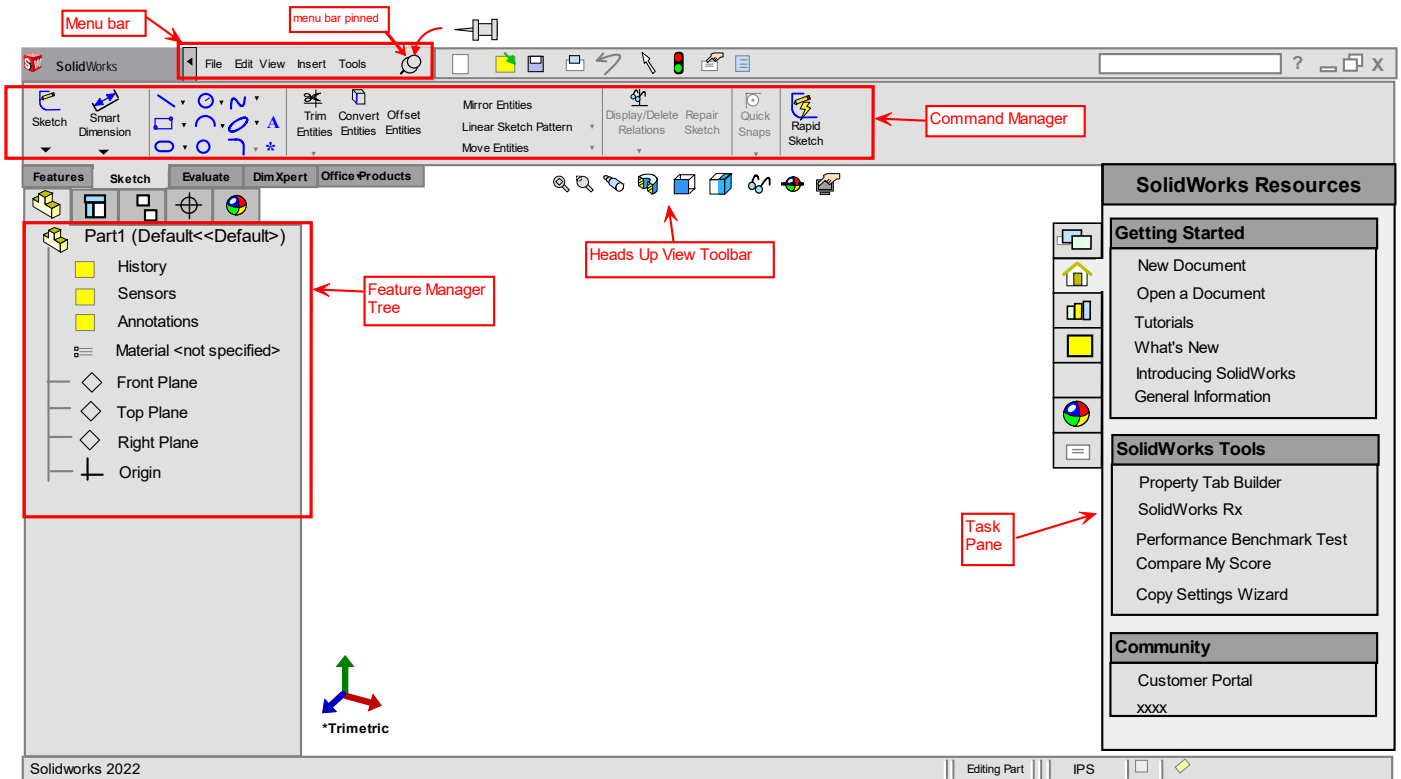
## CHAP 1 - INTRO



If you missed the units and dimension standards at this initial window they may be changed later in the DOCUMENT PROPERTIES.

Set the SW Dimension Standard to ANSI, the standard for the US. Note that this unfortunately does not translate to the SW templates. We must manually make those changes in the "Document Properties". See SOLIDWORK CUSTOMIZATION in a later chapter. It is a good idea to create a custom set of templates right away.

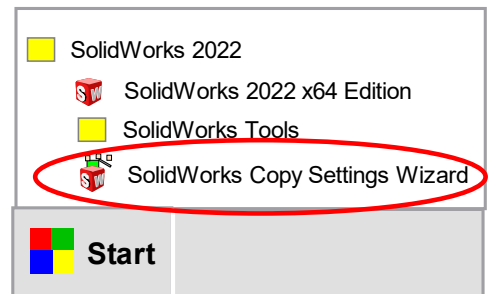
### 1.3 SW Interface



### 1.4 SW Settings

There are 2 basic SW settings:

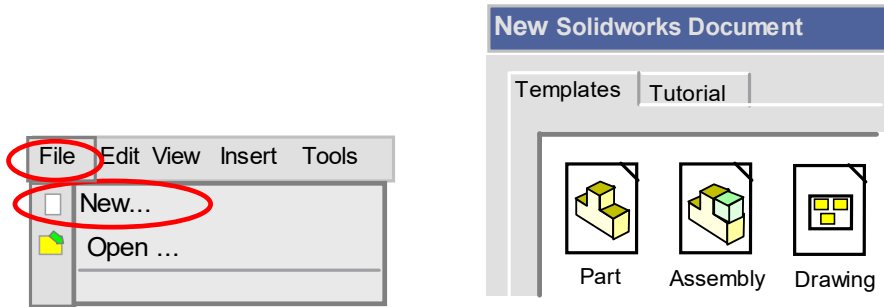
1. System options - settings tied to the SW installation on the computer
2. Document properties - settings tied to templates (one each for part, assembly, & drawing). Changes made here only apply to the CURRENT document. If you don't want to repeat the changes on the next document, you must alter or create a new template. (instructions below).



Settings may be manually set or imported (if you have desired settings saved to a file).

## 1.5 Opening New Part

Open a New Part (File > New > Part > OK).



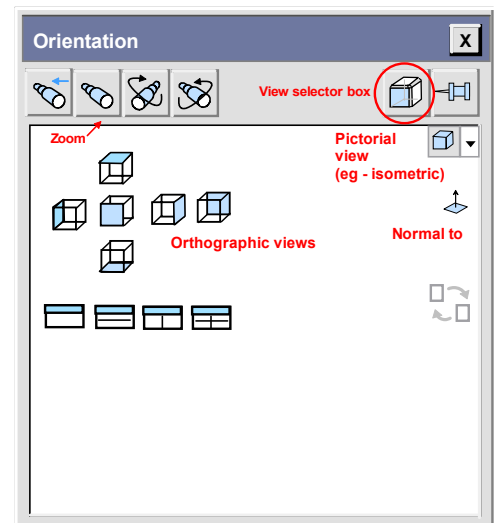
## 1.6 Basic Controls

In part and assembly mode, you must be able to pan, zoom, and rotate the model.

- Rotate view – push scroll-button down and drag
- Pan – push scroll-button down, hit CTRL key, and drag
- Zoom – roll the scroll wheel

View Orientation Changes (press SPACE BAR)

- View selector (the glass box) – turn off by clicking
- Orthographic views (front, top, right, etc.)
- Isometric
- Normal to



After pressing SPACE, the SW "View Selector" Cube will appear. You can use the View Cube by selecting one of its faces to re-orient the view of your part. You can turn off the View Cube in the dialog box.

## 1.7 Command Manager

The command manager is generally essential and should appear. If it does not you can get it to show using the following steps

Tools > Customize > Toolbars (tab) > [x] Enable CommandManager

## 1.8 Printing

You are most likely going to only print drawings. And you are most likely to print 2 ways:

1. Directly to a printer
2. To PDF file.

A pdf file can be printed on almost any computer. The computer does not need SW installed.

However you will generally want all lines to print BLACK. SW encodes DRIVEN dimensions as grey, but you don't want those printing grey. Here are some ways to get that done.

#### STEPS printing from SW to a printer

1. Print (File > Print, or ctl-P)
2. Under printer properties select (x) Black and White
3. Press OK to print

#### STEPS printing to PDF

1. Change the display of driven dimensions  
(Properties > System Options > Colors > Dimensions Driven > (set to BLACK))
2. Print to pdf
3. You may wish to reset that Driven-color dimension back to grey.  
(note - System Options are tied to the installation on your computer, not to templates, so once set, you should be good).

# \*\*\*\*\* SKETCHING \*\*\*\*\*

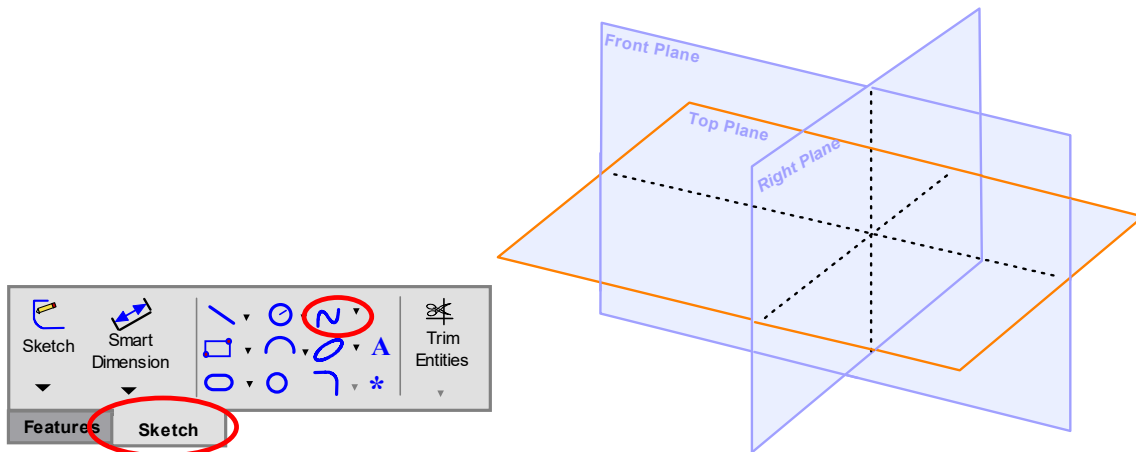
## CHAP 2 - SKETCHING


Sketches are 2-dimensional (2D) graphical entities drawn on flat surfaces. Sketches are often the first step to producing 3D geometry.

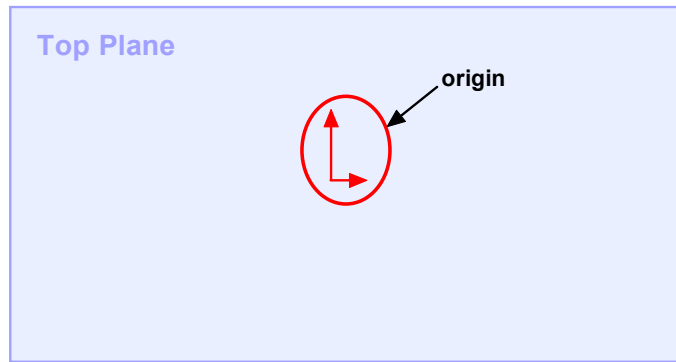
### 2.1 Sketching First Steps

To create a sketch click the "Sketch" tab in the command manager. There you will see a variety of 2D entities to select (lines, arcs, circles, splines, etc.). Select an entity. You will be presented with 3 default planes: Front, Right, and Top.

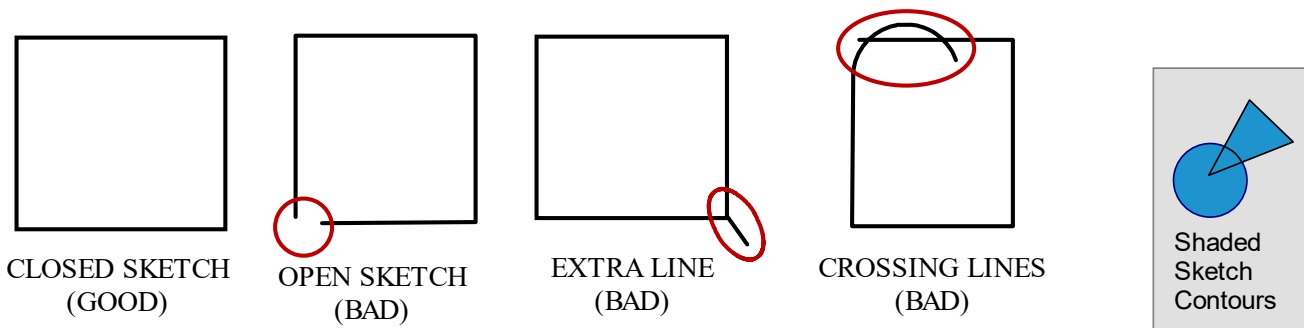
Sketches are drawn on flat surfaces. At this point, there is not part geometry to draw on, so most users will sketch on one these default planes. Or you can create a plane to draw on. Pick one (it highlights orange) to draw on it.



It is easiest to sketch with the sketch plane "normal to" your viewing perspective. To change this, press the space bar to get the ORIENTATION DIALOG BOX. Select the NORMAL TO icon . The 2 red arrows are the sketch origin.



You can sketch a shape with any combination of lines, arc, and splines. However, the sketch must be CLOSED if it is to be used to create a solid. It cannot be OPEN, have extraneous lines, or crossing lines. Use the REPAIR SKETCH feature if you can't find your error. SW 2017 introduced "Shaded Sketch Contours" where a properly closed sketch would be shaded. This can be turned off as shown.



Because of the parametric nature of SW it is often a good idea just to get the general shape/geometry correct and worry about the exact dimensions later since these can be changed.

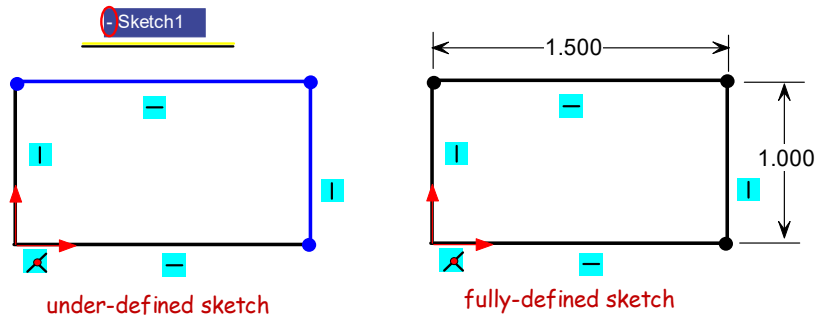
## 2.2 Fully-Defined Sketches

A SW user will generally use a variety of sketch entities to complete the geometry of a sketch (lines, arcs, etc.). Best practices are to make the sketch FULLY DEFINED. This means that the user specifies all of the DIMENSIONS and GEOMETRIC CONSTRAINTS to fully constrain the sketch, making it completely un-ambiguous. This means the sketch has no missing dimensions or information. A fully defined sketch appears completely black. Plus "fully defined" will show in the lower right of the SW window.



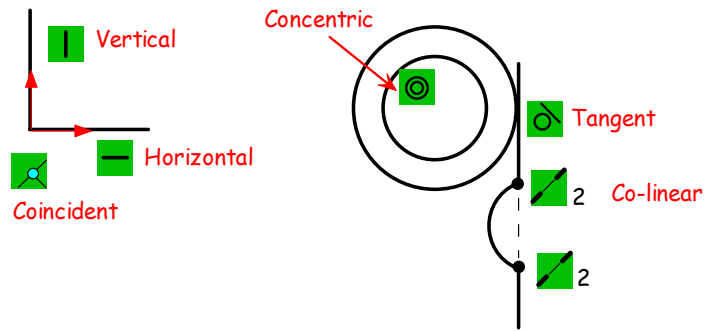
In contrast, an UNDER-DEFINED sketch has missing information (dimensions, geometric constraints, etc.). Thus, the sketch is ambiguous. An under-defined sketch shows BLUE and the sketch feature shows a (-) in the feature tree. It is possible to reveal the missing constraint by selecting portions of the sketch and dragging. This can be tricky though as you must grab the correct item.





### 2.3 Sketch Constraints and Relations

SKETCH RELATIONS are the little green boxes that indicate geometric constraints or relations like: HORIZONTAL, VERTICAL, TANGENT, COINCIDENT, CONCENTRIC, CO-LINEAR.

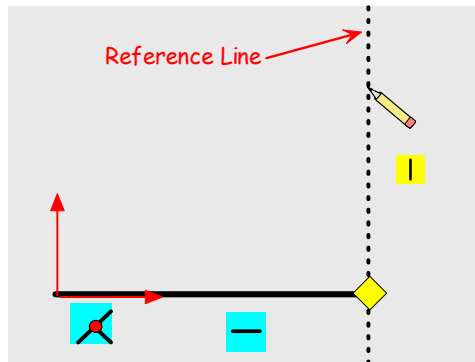


### 2.4 Chain of Lines

Create a chain of lines by selecting the line command and then just left click, move mouse, L click, move mouse, etc.



### 2.5 Reference Lines



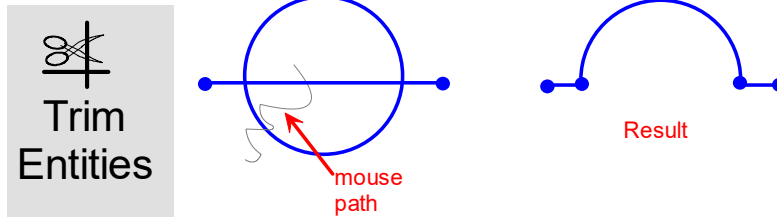
Solidworks will present reference lines while you are sketching. These are intended to assign various geometric and alignment constraints (like AutoCAD "OSNAPS"). Suppress by pressing "Ctrl" key.

### 2.6 Geometric Constraints

- Concentric – arcs and circles share same center
- Co-linear – 2 lines lie on the same infinitely-long "line"
- Perpendicular – 2 lines that are 90 degrees rotated from each other
- Co-incident – share the same point
- Sometimes used for co-planar, co-linear

## 2.7 Power Trim

Select the power trim button and drag the mouse (L click) to move the cutting cursor.



## 2.8 Splines

Splines are smooth curves that are created that pass through the points you select.  
Double click the last desired point.  
Press the ESC key if you clicked one point too far.

## 2.9 Fully Define Complex Sketch

(Tools > Dimensions > Fully Define Sketch)

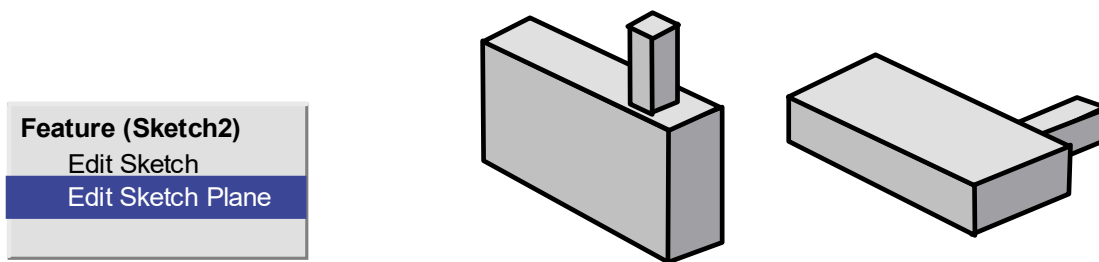
Sometimes a complex sketch can be tricky to fully define.  
Solidworks can automatically define the sketch (i.e., add dimensions)

# \*\*\*\*\* BASE & SKETCH-BASED FEATURES \*\*\*\*\*

## CHAP 3 - BASE & SKETCH-BASED FEATURES

### 3.1 Base Feature

The base feature in SW is the first feature, or starting point. It must be created with a sketch. The orientation of the base feature may NOT be changed if you dimension to the default planes (front, right, top). Thus, sometimes it is best to create a plane and draw on IT rather than one of the default planes.



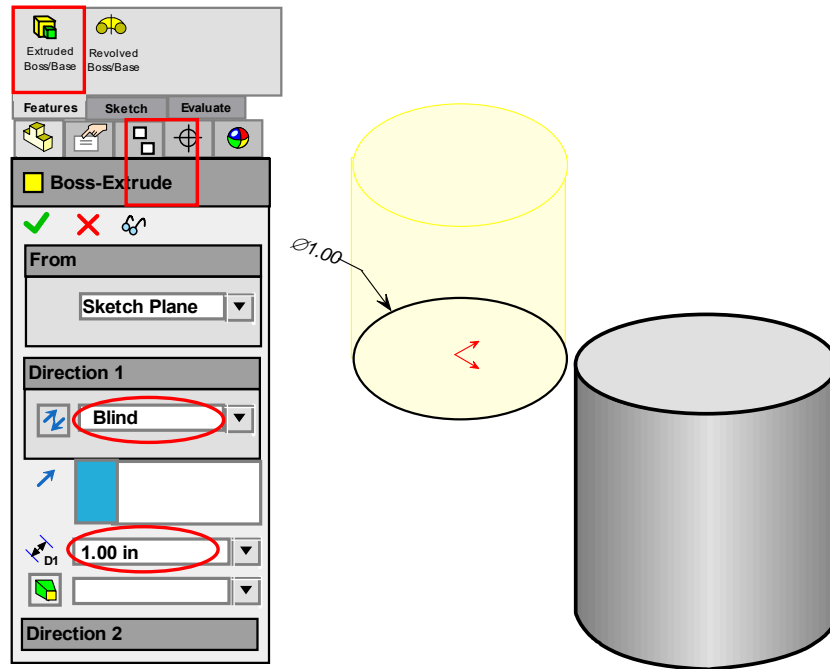
### 3.2 Sketch-Based Features

Generally we can think of these functions as taking 2D geometry (the sketch) and creating 3D geometry from it. There are 6 basic ways of creating sketch-based features.

1. Extruded boss/base
2. Revolved boss/base
3. Extruded cut
4. Revolved cut
5. Sweeps
6. Lofts

#### 3.2.1 - Extruded Boss/Base

Extruded boss/base takes a CLOSED sketch and EXTRUDES it into a solid, generating material. Generally you draw a closed 2D sketch and extrude along a line perpendicular to that sketch, thus creating 3D geometry.



## STEPS

1. Create a CLOSED sketch
2. Pick Extrude Boss/Base (Features > Extrude Boss/Base)
  - you get a yellow preview if you are ok
3. Pick the end condition (Direction 1 > xxx) and distance
4. Click OK (green check)

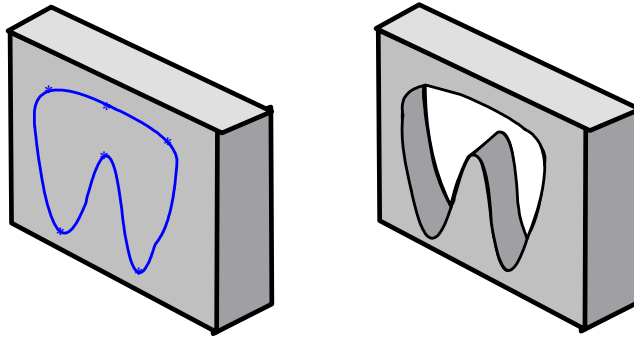
The resulting extrusion depth is BLUE since it's not defined in the sketch but instead in the extrude feature parameters.

### 3.2.2 - Extruded Cut

Extruded cut is much like extruded boss/base but instead of creating solid material, it removes it. It takes a CLOSED sketch and EXTRUDES cuts (removes) material from a solid. Generally you draw a closed 2D sketch and extrude along a line perpendicular to that sketch, removing material from a previously-existing solid.

## STEPS

1. Create a CLOSED sketch
2. Pick Extrude Cut (Features > Extrude Cut)
  - you get a yellow preview if you are okay.
3. Pick the end condition (Direction 1 > xxx) and distance.
4. Click OK (green check)



### 3.2.3 - Revolved Boss/Base

Revolved boss/base takes a CLOSED sketch and REVOLVES that about an axis into a 3D solid. The sketch must have an axis (or line) and a solid sketch on ONE side of that line.

#### STEPS

1. Create an axis and a CLOSED sketch that lies entirely on one side of that axis.
2. Pick Revolved Boss/Base (Features > Revolved Boss/Base)
  - you get a yellow preview if you are okay.
3. Pick the end condition (Direction 1 > xxx) and angle.
4. Click OK (green check)

### 3.2.4 - Revolved Cut

Revolved cut is much like revolved boss/base but instead of creating material it removes it. It takes a CLOSED sketch and REVOLVE cuts that from an existing solid.

The sketch must have an axis (or line) and a solid sketch on ONE side of that line.

#### STEPS

1. Create an axis and a CLOSED sketch that lies entirely on one side of that axis.
2. Pick Revolved Cut (Features > Revolved Cut)
  - you get a yellow preview if you are okay.
3. Pick the end condition (Direction 1 > xxx) and angle.
4. Click OK (green check)

### 3.2.5 - Sweeps

A SWEEP is an extrusion of a PROFILE along a curved PATH.

Generally you need a closed sketch and a guide curve that usually starts perpendicular to the closed sketch.

This feature may be used for things like tubular features (like in a min-baja design).

#### STEPS

1. Create a closed sketch and also a 3D line.
2. Pick Sweep (Features > Sweep)
  - you get a yellow preview if you are okay.

3. Pick the end condition (Direction 1 > xxx) and angle. ????????????????
4. Click OK (green check)

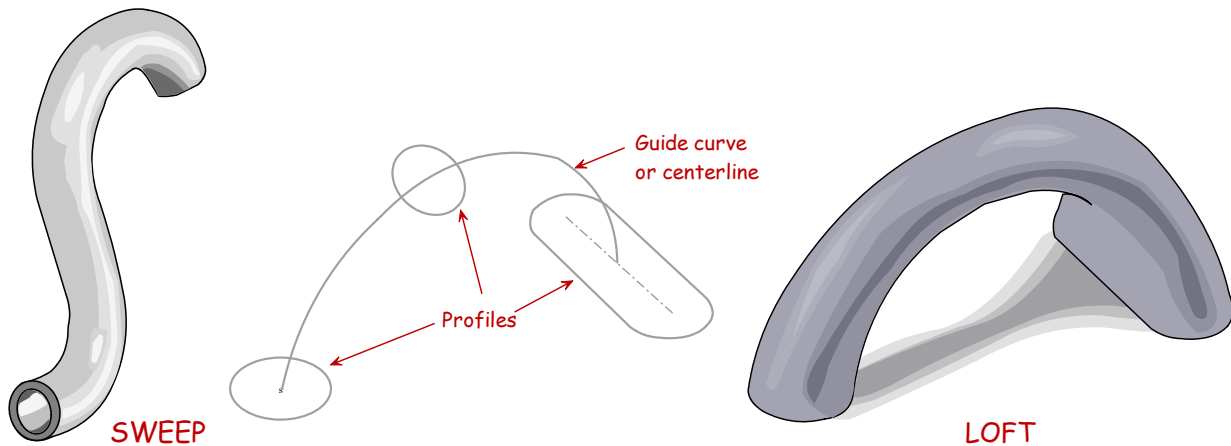
### 3.2.6 - Lofts

A LOFT is also much like a sweep but shape is based on multiple close sketches (profiles) that are generally different shapes and a guide curve the lives in 3D space.

This feature may be used for things like tubular features (like in a min-baja design).

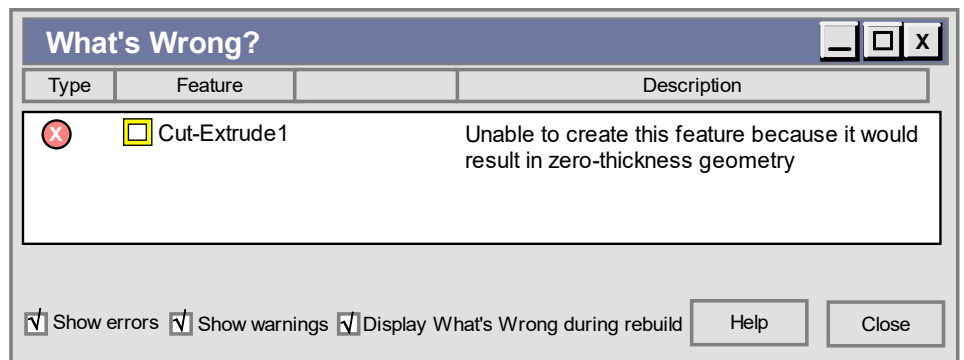
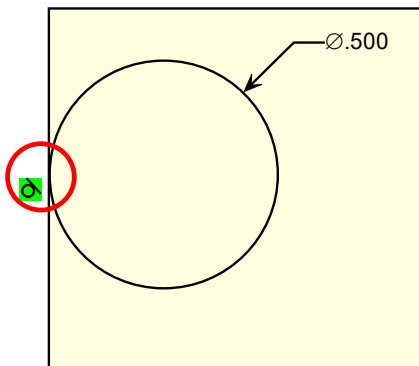
#### STEPS

1. Create a closed sketch and also a 3D line.
2. Pick Sweep (Features > Sweep)
  - you get a yellow preview if you are okay.
3. Pick the end condition (Direction 1 > xxx) and angle. ????????????????
4. Click OK (green check)



### 3.3 Zero-Thickness Solids

Zero-thickness solids are considered impossible geometries in SW. If you sketch a circle that tangent to a line and try to extrude it, you will get an error. This error can also occur with other constructs.



# \*\*\*\*\* PART DIMENSIONING \*\*\*\*\*

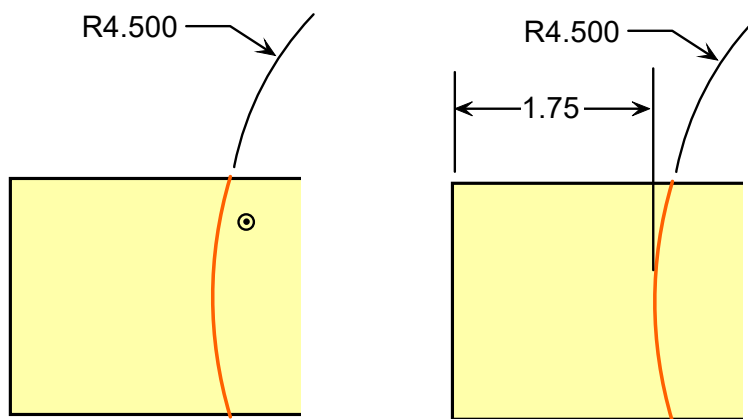
## CHAP 4 - PART DIMENSIONING

Dimensioning in SW is "smart". It creates the desired dimension based on the context of the items you select.

### 4.1 Arc Conditions

You can dimension to different ARC conditions. Press SHIFT with the cursor in the following positions.

1. Minimum - between the 90 degree window
2. Center - outside the 90 degree window
3. Maximum - moved to the imaginary 90-degree window (on far side of circle)



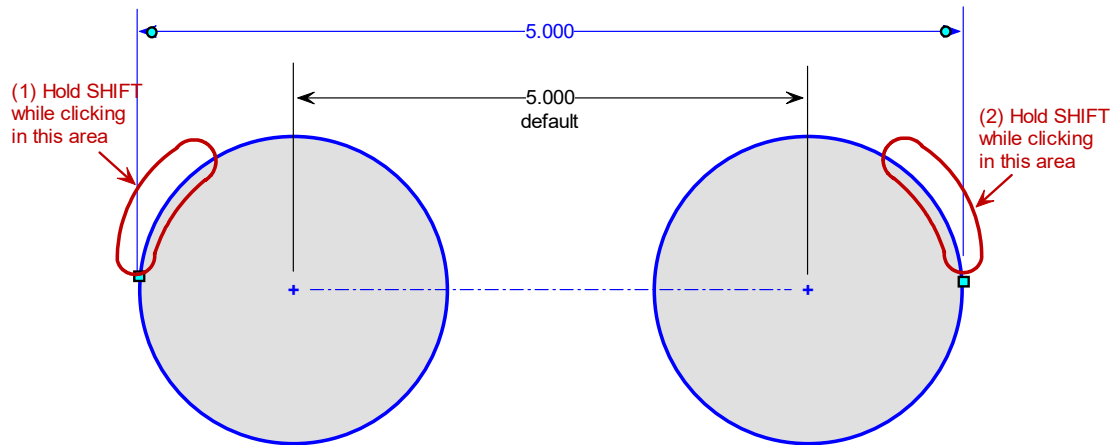
## 4.2 Circle or Slot Dimension Options

Simply clicking the 2 circles will produce a center-to-center dimension.

To dimension to the outside or inside of the circle hold the SHIFT key while selecting.

Select by clicking NEAR the area of circle you want to dimension to. For example, click near the outer area to dimension to the outside of the circle.

You can dimension from outer to center by holding shift while selecting the outer part of the circle, then releasing shift and clicking the second circle.



\*\*\*\*\* PARTS \*\*\*\*\*

## CHAP 5 - PARTS

A part is a single object in SW. A part is the combination of a sequence of discrete features. All of the features and their sequence are summarized in the FEATURE MANAGER TREE.

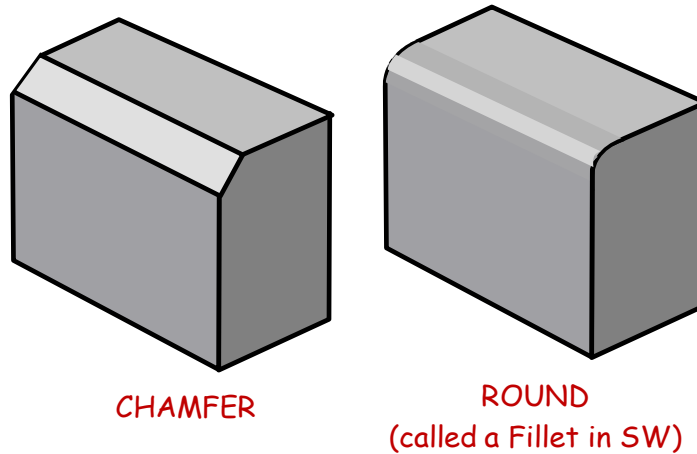
### 5.1 Chamfers

Chamfers are beveled edges.



## 5.2 Fillets

Fillets are rounded edges. Fillets can be problematic with intersecting edges.



## 5.3 Hole Wizard

### STEPS

Select a surface FIRST

Use the hole wizard (don't just draw a circle)

Different types (counterbore, counter-sink)

Add a cosmetic thread (gives the dashed lines) w/o callout

Place second hole

Add relations – horizontal

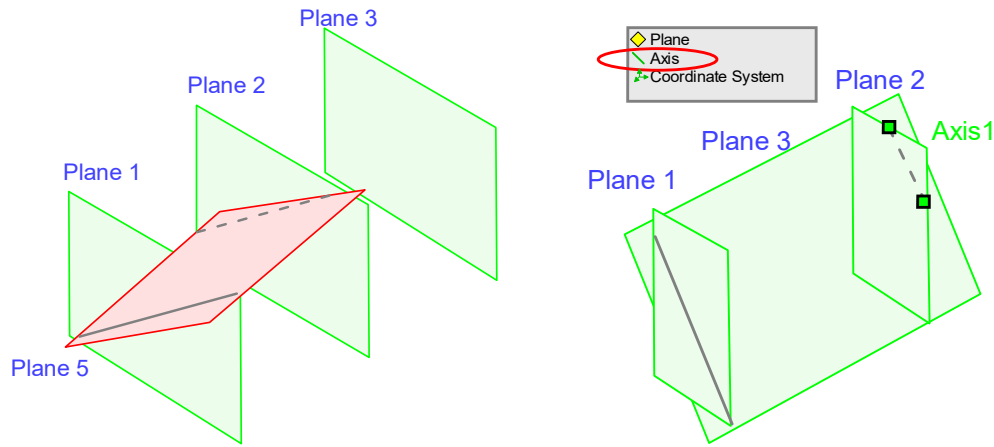
Is it horizontal or vertical (use pre-defined views, or look at orientation of dimension callouts)

## 5.4 Reference Geometry

Sometimes we create reference geometry (planes, surfaces, lines, points) to aid in our creation of part geometry.

Create offset planes, then create a plane thru line & pt

Pick axis (intersection of 2 planes, pick 2 planes). Gives intersection



## \*\*\*\*\* ASSEMBLIES \*\*\*\*\*

### CHAP 6 - ASSEMBLIES

Assemblies are produced when you assemble multiple parts together. Assembly occurs by applying GEOMETRIC CONSTRAINTS which are called MATES. An assembly is also summarized in the FEATURE MANAGER TREE as a sequence of assembled parts and their mates.

#### 6.1 Starting an Assembly

In "bottom-up" design, assemblies are generally created starting with the parts.  
 First open the part that will be the first part of the assembly. Make sure the part window is the active one.  
 Go to File > Make assembly from part.  
 Check the origin.

Bring in (check graphics view)  
 Place the part so that the origin of the assembly is coincident with the part's origin (you see the double arrows). Helps to have origins showing (View > Hide/Show). Can hide origins after inserting first part.

In the FEATURE TREE you should see an "(f)" next to the part (where (f) = fixed).

Note you can unfix using "FLOAT". Unfixed parts show "(-)"

Window > Tile Horizontally (so you can see all the open parts).  
 Pick the second part and drag it into the assembly. (or use Assembly tab > Insert Part)  
 It helps to rotate the part close to the correct orientation.

## 6.2 Mates

"Mates" is the term for the GEOMETRIC CONSTRAINTS that fit one part to another one. Inserting a pin into a hole would be a mate of CONCENTRICITY. Mates may result in under-constraint, which would result in movement of the assembly. This is a really cool feature in SW. Assemblies can be machines that move (like an internal combustion engine). That's why this type of program is essential for mechanical engineers.

It is very easy to create constraints that conflict with each other, or ones that over-constrain the assembly. This requires a lot of thought to resolve.

COINCIDENT

Has many meanings

CONCENTRIC

CO-PLANAR

GEAR MATES

These will sometimes work if you do it right. Eg, if you spin the screw, the jaw moves.

DISTANCE

## 6.3 Explodes

Assemblies can hide components, and they can be difficult to see how the parts fit together. One technique for addressing this is to EXPLODE the assembly. You may have seen this in instructions for assembling a piece of furniture. SW constrains one part to another one. Inserting a pin into a hole would be a mate of CONCENTRICITY. Mates may result in under-constraint, which would result in movement of the assembly. This is a really cool feature in SW. Assemblies can be machines that move (like an internal

STEPS

1. Click part
2. Click axes, apply
3. Done
4. Explode line sketch
5. Click on screw
6. Click

## 6.4 Adjusting Parts Based on Assembly

Bottom-up design is almost always taught and learned first. It's easier to comprehend and easier to form the assembly. However, the challenge of bottom-up design is making the small dimensional changes at the part

level so the assembly comes together as desired. Often you will find yourself tweaking part dimensions until something lines up as desired.

It may be helpful to add a dimension to the assembly to help aid you in determining how much to change a part dimension to obtain some kind of alignment.

## \*\*\*\*\* DRAWINGS \*\*\*\*\*

### CHAP 7 - DRAWINGS

Drawings may be produced from individual parts or of assemblies. Drawings also have their own version of the FEATURE MANAGER TREE. In the case of drawings, the tree defines the drawing as dependent on a part and the various created views of the part.

#### 7.1 Sheet Formats

Sheet formats contain the border, title block, and other settings.

##### EDIT SHEET FORMAT

(RH click white space > Edit Sheet Format)

Text in sheet format is only editable while in "Edit Sheet Format" (like paperspace in AutoCAD).

##### OTHER ITEMS

###### Part

File > Properties > Custom (e.g. – metal 6061-T6)

###### Drawing

Edit sheet format

Highlight field

Part name – link to file name

Notes – view dependent vs. independent

###### Sections

Draw the line first (align the holes, view/temp axes)

###### Creating Fields (File > Properties > Custom)

Make field

Highlight letters

Click button

Props/ custom

Student text 0000000

Or views from drawing

Insert model  
Highlight model  
Choose first view (front)

## 7.2 Drawing Scale

(RH click white space > Drawing scale).

## 7.3 Detail Views

(Insert > Drawing View > Detail)

Drag out a circle. Adjust its center point  
Set style:

Per Standard

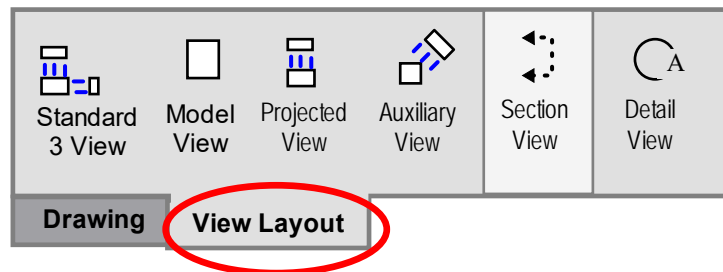
Broken Circle (use this)

Set scale (e.g., 4:1)

Detail view includes a label: "DETAIL A: SCALE 4:1" (this can be deleted if desired)

Detail view can be moved to desired position on drawing

## 7.4 Section Views



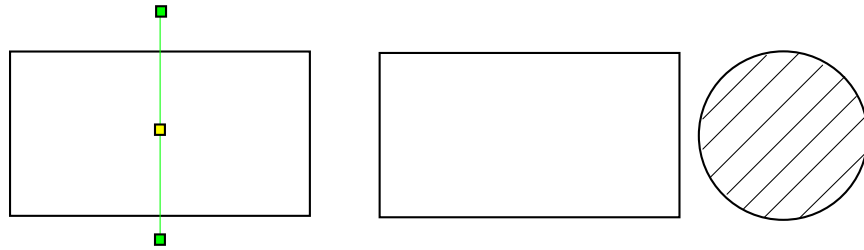
Draw the line first.

Then pick on section view (make sure it's 3rd angle standard)

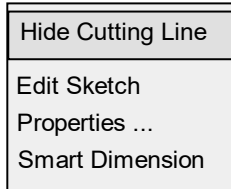
## 7.5 Revolved Sections

STEPS

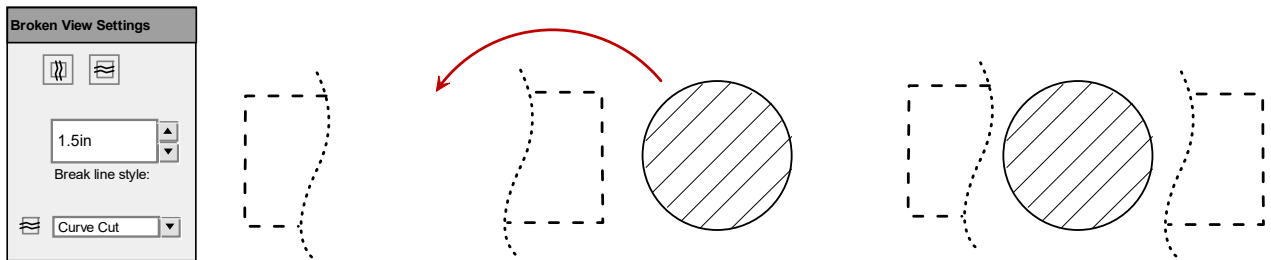
1. Place view
2. Draw line on the view



3. Use line to create a section
4. Remove the "Section A-A" notation (select and delete)
5. Right click the cutting-plane line > Hide

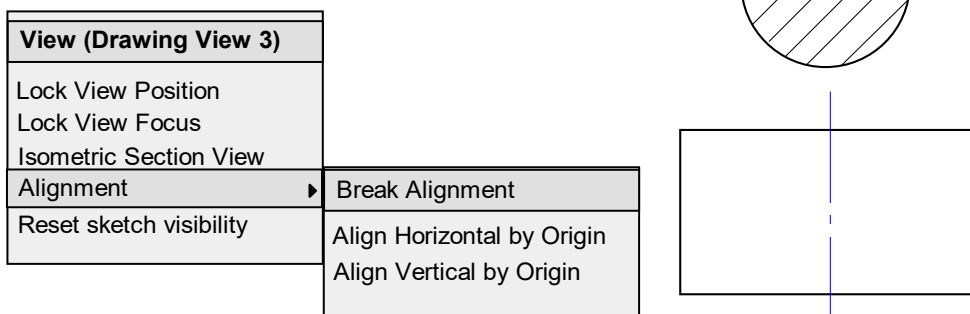


6. Break the view – set the gap and line style
7. Move the section into the break area



## 7.6 Removed Sections

1. Create the section as usual.
2. Break the alignment (RH click section view > Alignment > Break Alignment)
3. Move the section view to the desired location



## 7.7 Half or Offset Sections

1. Create the section as usual.

- B. Aligned sections – draw a kinked line first. Aligns to the 2nd (1st?) line selected.
- C. Spoke/rib rules – SW has no clean way of taking out the hatching. Must draw
  - 1. Sketch 2 boundary regions. Then create an Area Hatch/Fill. Use Line Format menu to change the color of the fill to white.

## 7.8 Auxiliary Views

Draw the line first then select the auxiliary view command.  
Click to place the view.

SECONDARY AUXILIARY VIEWS

## 7.9 Drawing Dimensions

AUTOMATIC DIMENSIONING

(Insert > Model Items > Dimensions)

SW can automatically insert dimensions based on dimensions specified in the part. Note the dimension will not appear if you don't have the proper view already inserted. Dimensions inserted are DRIVING DIMENSIONS, which can alter the part. Many companies only allow DRIVEN dimensions to appear on drawings. You can convert these dimensions to be driven.

Select dimension  
(Properties > Other (tab))

MANUAL DIMENSIONING

(Smart Dimension)

Manual dimensioning inserts driven dimensions which appear gray (this can be changed in settings). When printing, select "Black and White" (or dims will be in grey).

### 7.9.1 - Radius vs. Diameter

You can switch between radius and diameter

RH click dimension > Displays Options > Display as Radius (or Diameter)

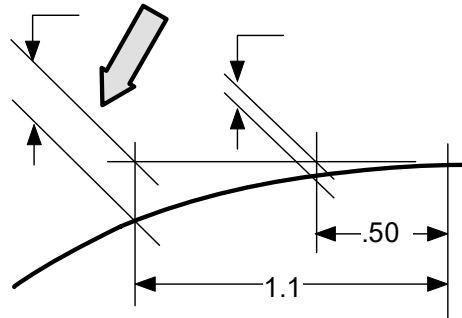
### 7.9.2 - Spherical dimensions

You must simply add an "S" to the dimension.

Select the dimension. In the dimension options > Dimension Text > (add an S before the items there)

## 7.10 Drawing Features

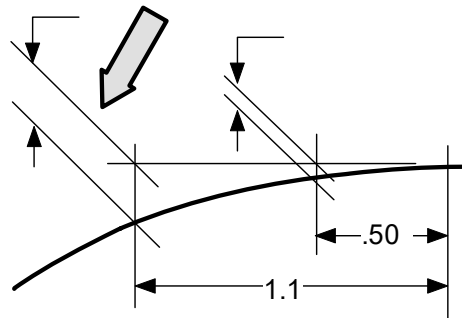
### 7.10.1 - Oblique Extension Lines



#### STEPS

1. Click the dimension
2. Grab the extension line grip
3. Drag

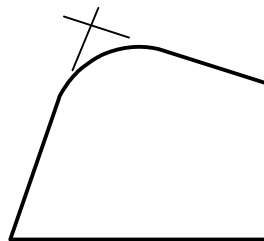
### 7.10.2 - Jogged Extension Lines



#### STEPS

1. Right click extension line
2. Display options > Jog

### 7.10.3 - Virtual Sharps



#### STEPS

1. Pick one line.
2. Hold Ctrl key and pick 2nd line.
3. Pick the point (\*)

### 7.10.4 - Arcs



Select the start point of the arc.  
Select the end point of the arc.  
Select a point between the 2.  
Place the dimension.

### 7.10.5 - Chamfers

RH click the 45, then the edge, then pull out the dim

### 7.10.6 - Centerlines

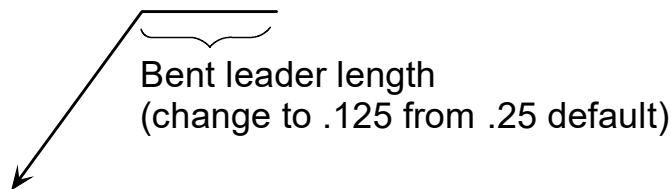
CL – don't use  
Dim CL – pick target symbol

### 7.10.7 - Hole callout

Must have used the Hole Wizard

### 7.10.8 - Symmetrical centerlines

Draw short fat lines on your own (workaround)  
Screw  
Relative view  
RH click sheet/ drawing views/ rel views  
Bent leader length



View/ temp axis (get \*)  
Edit – RHC line  
Offset  
Select this  
Use templates

## 7.11 Title Block Fields

The fields in a SW sheet format are parametric, and so you don't alter them the way you might think.  
Alter them by entering:

(File > Properties > Sheet Format).

## 7.12 Assembly Drawings

Drawing

Import 2 of them (will convert configur of 1)  
Props/ uncheck show exploded state  
Hide all (temp axis, origins)  
Balloons  
Formatting them  
BOM  
Edit sheet format  
Click upper left, RHclick, set as anchor, BOM  
Back to sheet  
    RH click  
    Table / BOM  
    Anchor (upper right)  
    Change col width – just drag  
Part  
    Props  
    Config specific  
    Add/change

## 7.13 Printing

Generally you will want to print in BLACK  
(File > Print > Print options > Color (set to Black and White))

\*\*\*\*\* **ADVANCED TOPICS** \*\*\*\*\*

## CHAP 8 - ADVANCED TOPICS

### 8.1 Freeform Surface Modeling

- I. Freeform (Insert > Surface > Freeform)
  - A. Step
    1. Pick surface
    2. Add curves > LH click to add
    3. Flip Direction (tab)
    4. Add curves > LH click to add
    5. Add points > LH click to add points
    6. Unclick add points, click a curve, select point and drag

## 8.2 Gears

### STEPS

1. Draw root, base, pitch, and outside circles (use Excel table to get the diameter values).
2. Set them as construction
3. Draw radial lines to base circle - horizontal, half angular pitch, full angular pitch (e.g., 4.5 & 9 deg)
4. Draw radial line that is 2X pitch (18 deg)
5. Draw involute as equation-driven curve
6. Set up equations (tools > equations > global variables)
7. Diametral pitch (P), number of teeth (n), pressure angle (phi), addendum (a), dedendum (d)

Clearance, c (b-a)

7. Pitch diameter, PD
8. Base circle dia,
9. When doing the equation-driven curve, must reference the dimension
  - a. Can't reference global variables

I was previously able to create a gear model with INVOLUTE shaped teeth. This requires creating EQUATION-BASED CURVES. Years ago I had trouble mirroring the curve but was eventually able to do it by copying the curve, mirroring it, then mirroring it again. In SW 2022 I tried to duplicate this but could not. The mirrored curve was not parametric. This I could not solve. There is an option to simply pull in a gear from SW mechanical components library. This did not work either. The model was not parametric.

I solved the problem by estimating the involute with a 3-point curve. This works well enough and the gear model is changeable.

## 8.3 Categories

1. Fractional
2. Props/ use doc units (uncheck)/ units
3. Type in as "5/16"
4. Units
  1. 50 mm
  2. Converting – tools/ option/ doc props

## 8.4 Tolerancing

Bilateral - for  $xx \pm xx$

### STEPS

1. Click dim, switch between dia (xxx) or depth (xxx) (confusing!)

## 8.5 Fillet

CONSTANT WIDTH

- Constant width (looks nice)

VARIABLE RADIUS

- Suppress already-done features
- Variable radius
- Nodes – eg. 5 (different radius value at each node)
- Set all

#### FACE FILLET

- Face fillet
- face fillet 2

Surface 1, surf 2, and hold line

Aesthetics!

With variable thickness part

Fill in

Body fillet

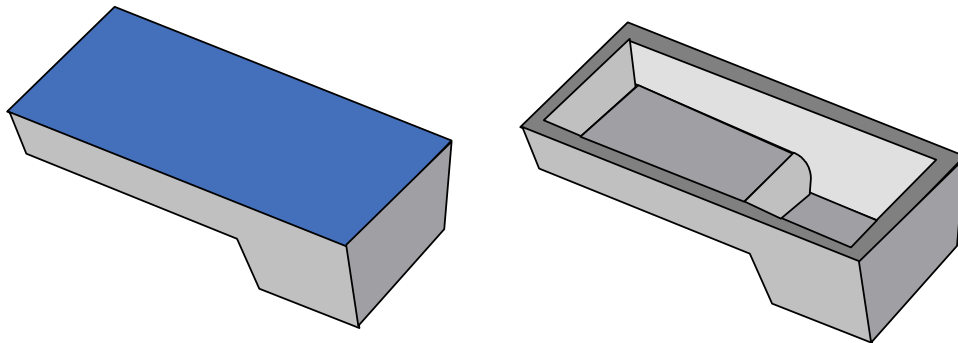
#### FULL ROUND

(select 3 surfaces in right order)

### 8.6 Surface Cut

A SURFACE CUT allows you to cut a part using a surface (flat or not flat).  
(Surfaces > Cut with Surface).

### 8.7 Shell



The SHELL feature takes a solid and hollows it out. The resulting "shell" part is often the type of part that may be injection molded.

(Features > Shell)

Then select a shell thickness.

Then select a surface (the side that the shell is OPEN TO).

### 8.8 Wraps

Tools/ customize

Font – pts (auto changes units and space)

Insert/features/wrap

How to

- Select surface
- Emboss – sticks out
- Deboss – goes in
- Scribe – written on

## 8.9 Equations

Relates different dimensions

STEPS

1. Select dimension
2. Tools/ equations
3. "Dimension Selected" = xxx
4. Result: dependent equations have a "sigma" ( $\Sigma$ ) in front

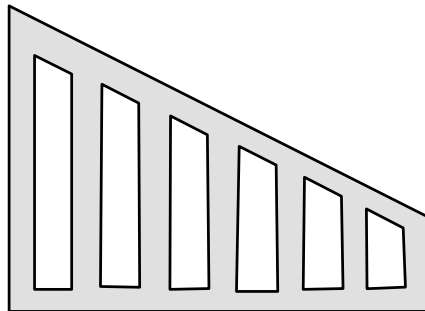
Configuration tables

1. Dimensions for part drawn from table (Rob may not cover)

Linking

1. Pick dim
2. RHclick/ link values
3. Share values (give dim a name, eg "top width")
4. Pick a 2nd dim
5. Neither controls
6. Global to part

## 8.10 "Vary Sketch" Patterns



Instead of a linear or circular pattern, a VARY SKETCH pattern creates a pattern based on a sketch. This is basically done when a DIMENSION is used for the pattern direction. Only then does the "vary sketch" option become available.

Create feature to be patterned

Linear Pattern

1. Direction 1  
For direction, open the feature pick the dimension that offsets the feature from the side wall  
Do NOT pick the lower edge (which is what we would normally do)  
Pick distance and number of instances.
2. Under Options > check "Vary sketch". Preview still does not look right.

3. Click "Full preview" (instead of Partial Preview) and it will look right.

a. Can create a sketch (don't create a feature from it), then use it later

## 8.11 Saving image

- A. Do from drawing mode (on blank format), doesn't work fm dwg mode
- B. Save as jpeg
- C. Adjust DPI
- D. Import to MS Word, crop, compress

## 8.12 GD&T

Datums

Basic dimensions – by double clicking dimension

Feature control frame

## 8.13 Surfaces

SW surfaces have no thickness or mass. Often they are used as construction geometry or references for creating part material.

Insert/face/delete – you go from 1 solid body to 1 surface body

## 8.14 Top-down assembly

Assembly is the parent

2 spheres (concentric)

Remove inner sphere – so you get a cavity

Split

- 1. insert/feature/split
- 2. hide vs. show
- 3. create new parts

E. Make new assembly

F. Make part in assembly

G. Make a mold

H. Import door – move it up in the tree

I. Edit part (mold), edit sketch

PARTIAL TOP-DOWN

Create parts with tab/slot assembly. Must position slots so parts position properly. Example would be to create tabs on part, pull into assembly, position properly. Then create slot features on the plate using the tabs.

CREATING A SET OF HOLES

One small top-down example is creating a set of mounting holes on a base plate for - for instance - an Arduino circuit board. The Arduino board is already modeled with proper hole spacing. One option is to get into the base plate part and match the dimensions on the Arduino part. This works but is a pain. A second option is to create an assembly that includes the base plate and Arduino. Mate the Arduino to the base plate as desired (onto surface, distance from 2 edges). Then make holes in the plate using the following steps:

1. Edit Part > Select base plate
2. Sketch on plate surface
3. Create a set of circles that are concentric with Arduino holes (may need "temporary axes" shown)
4. Size the holes
5. Extrude cut the holes
6. Exit the "Edit Part" mode

The part will have the mounting holes now, even if you just open the base plate part alone.

## 8.15 Extrude to surface

- IV. Create a container (fm 1 part)
  1. Create a "cup" type part (surface)
  2. Create 2 configurations (inner thicken, outer thicken)

## 8.16 Extrude to Multiple Surfaces

Must KNIT surfaces first (Surfaces menu > Knit surface)

## 8.17 Over mold

STEPS

1. Cut sweep
2. Select ALL surfaces
3. Offset plane
4. Trim – also extends (corner)

KNIT

Knit makes 2 surfaces one

VI. Insert/boss/base

VII. Display relations

VIII. Olive drab = a problem!

IX. Replace Face

## 8.18 Global variables (use a dummy sketch)

Create a dim on a "useless" sketch

Link/shared value – give name

Equations – click on sketch to get the dim

## 8.19 Mini baja – piping

### STEPS

1. Sketch offset plane
2. Select, select chain
3. Convert entities
4. Another sketch
5. Draw circle
6. Horizontal bar – extend sketch so extrusion goes thru
7. Extrude
8. Do first, sweeps can be disjoint
9. Mirror
10. Split
11. Do same with the top
12. Parts have the "fish mouth"
13. Make drawing
14. Put in centerlines
15. Can put pts at cl intersections to dimension
16. Shelling?

## 8.20 Ordinate dimensioning

Ordinate dimensioning is when all dimensions originate from a single point. This is a good technique for sheet metal where tolerances are not critical.

(RH click a view > More Dimensions > Ordinate)

Click the first edge.

Draw out dimension.

Click subsequent edges

## 8.21 Draft Angle

DRAFT ANGLE tapers one side of the part. This feature is often used for molded parts as the taper makes extraction from the mold easier.

### STEPS

1. Select draft
2. Pick neutral plane (doesn't change)
3. Pick faces to draft
4. Pick direction (arrow points in direction of convergence)
5. Or can draft during extrusion
6. Can draft just 1 side

## 8.22 Breaks



## STEPS

1. Highlight view
2. Insert/drawing view/vertical break
3. Position break lines
4. RH click view/break view
5. Adjust scale on sheet

## 8.23 Ribs

### STEPS

1. Select plane
2. Select line
3. Rib (normal to, thickness)
4. Rib extrapolates to walls (stops at other rib if sketch is per right below)

### PARALLEL RIBS (ribs parallel to sketch)

1. Highlight sketch, press 'rib'
2. Parallel to, thickness
3. Will extrapolate (right pic)
4. Warning – sketch cannot go all the way to corners (or error)

## 8.24 Design table

### STEPS

1. Click dim/more props
2. Name... change it ("width@Sketch1") (can also name the sketch!)
3. Note – name your features in a useful way
4. Insert/design tables
5. RH click dim/props
6. Dim – get a table

## 8.25 Booleans

# \*\*\*\*\* SW CUSTOMIZATION \*\*\*\*\*

## CHAP 9 - SW CUSTOMIZATION

Once you have SW set up the way you like, you can save those settings to use on another computer. This is the COPY SETTINGS WIZARD. Once done, this saves a "swSettings.sldreg" file.

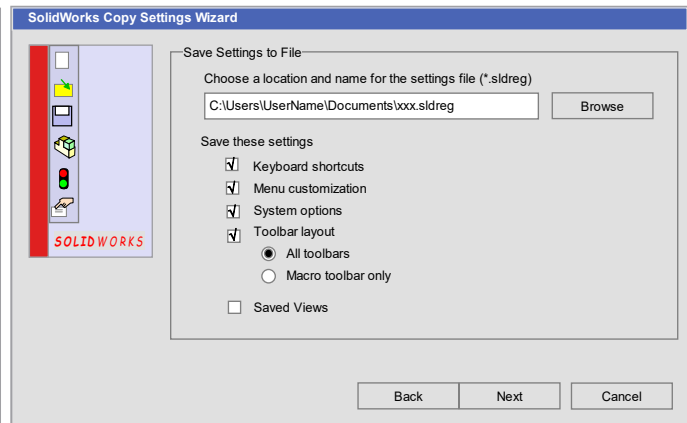
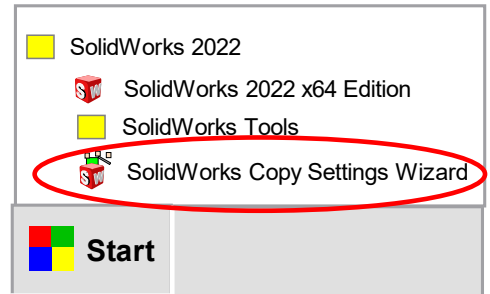
Go to: Start > All Programs > Solidworks 20xx > Solidworks Tools > Copy Settings Wizard 20xx

If you are on a new or reformatted computer, you can install SW and set up the settings either MANUALLY (see below) or IMPORTED using the COPY SETTINGS WIZARD.

Go to: Start > All Programs > Solidworks 20xx > Solidworks Tools > Copy Settings Wizard 20xx

It saves a swSettings.sldreg file. This file can then be used to import those SW settings to another computer. To IMPORT settings, follow the same steps but pick "Restore Settings".

Import settings – same steps as above but pick "Restore Settings".

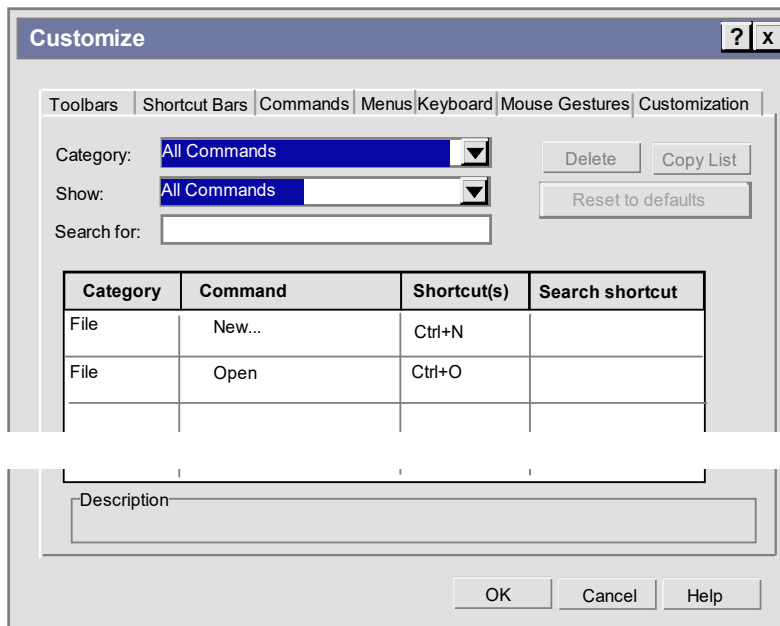


### 9.1 System Settings

System Settings – These options are tied to the SW installation on a computer, not to the template files. System Options (Tools > Options > System Options (tab))

Here are some settings that CT does (these are optional)

1. Display Style > Tangent Edges > Use Font (default is Visible)
2. Colors – Change the paper color to white (default is yellowish) (Colors > Drawings, Paper Color)



## 9.2 Keyboard Shortcuts

(Tools > Customize > Keyboard tab)

Keyboard shortcuts for view changes can really speed up your work. Note SW shows the keyboard letter in CAPS even though you'll enter in lower case.

CT's keyboard shortcuts (shortcuts work for both left or right side mousing)

View /Zoom to fit	(add J, F is default)
View/ Zoom to Area	(L, S... must remove L fm tool/line)
View / Zoom to selection	(K,D)
View / Rotate	(U, R)
View / Pan	(P)
Others/ Isometric	(I)
Insert/ mate	(M) (for assembly mode)
Tools/compon.rotate	(N) (for assembly mode)
Others/normal to	(O) (as in orthogonal)
Edit / Rebuild	(B)

## 9.3 Custom Toolbars

(View > Toolbars > )

The command manager "ribbon" along the top is standard. However, you can also add and customize toolbars.

## 9.4 Customized Templates

SW comes with a default set of part, assembly, and drawing templates. However you can create and customize your own.

A key setting to change is the DIMENSION STANDARDS. While we have set the dimension standard at the start of the installation process as ANSI, it does not seem to propagate to the SW templates. It is also suggested that you set the default display of dimension places at 3 (X.XXX).

There are 3 default templates in a specific file location. Those templates are:

Part.prtdot  
Assembly.asmdot  
Drawing.drwdot

You are free to open any of these templates, make changes, and then save those as NEW templates. It is suggested you do not over-write the default ones though. Save the new altered ones with a slightly different name in the same SW templates folder. They will then appear when you go to create a new part, assembly, or drawing.

### 9.4.1 - New Part Template

#### STEPS

1. Open a new part (File > New > Part)

Make settings changes below (Tools > Options > Document Properties)

2. Change Dimension Standard to ANSI for all items under [+] Dimensions (Tools > Options > Document Properties > Dimensions)  
Under each item XXX under dimensions, set "Base XXX dimension standard to ANSI),  
Where XXX is: angle, angle running, arc length, chamfer, diameters, hole callout, linear, ordinate, radius.
3. Dimensions display set to 3 places (Dimensions > Primary Precision)
4. Save as new template (e.g., Part\_AA.prtdot) (keep original template) in default directory (C:/ProgramData (hidden)/Solidworks/Solidworks20xx/templates)

The new template should now appear as an option when you make a new part.

### 9.4.2 - New Assembly Template

#### STEPS

1. Open a new assembly (File > New > Assembly)
2. Cancel out of any automatically-opening windows (part selection, etc.)

Make settings changes below (Tools > Options > Document Properties)

3. Change Dimension Standard to ANSI for all items under [+] Dimensions (Tools > Options > Document Properties > Dimensions)  
Under each item XXX under dimensions, set "Base XXX dimension standard to ANSI),  
Where XXX is: angle, angle running, arc length, chamfer, diameters, hole callout, linear, ordinate, radius.
4. Dimensions display set to 3 places (Dimensions > Primary Precision)
5. Display the origin (View > Hide/Show > Origins) (helpful when pulling in first part)
6. Save as new template (e.g., Assembly\_AA.asmdot) (keep original template) in default directory (C:/ProgramData (hidden)/Solidworks/Solidworks20xx/templates)

The new template should now appear as an option when you make a new assembly.

### 9.4.3 - New Drawing Template

#### STEPS

1. Open a new drawing (File > New > Drawing)
2. Cancel out of any automatically-opening windows (sheet format, views, etc.)

Make settings changes below (Tools > Options > Document Properties)

3. Change Dimension Standard to ANSI for all items under [+] Dimensions  
(Tools > Options > Document Properties > Dimensions)  
Under each item XXX under dimensions, set "Base XXX dimension standard to ANSI),  
Where XXX is: angle, angle running, arc length, chamfer, diameters, hole callout, linear, ordinate, radius.
4. Dimensions display set to 3 places (Dimensions > Primary Precision)
5. Make visible lines THICKER (Line Font > Visible Edges > 0.0138) (change from 0.0098)
6. Dimension Text - shrink it (Dimensions > Font > 0.100) (change from 0.125)  
Font - keep as Century Gothic (but Simplex is an option?)
7. Save as new template (Draw\_AA.drwdot) (instead of Drawing.drwdot) in default directory.  
(C:/ProgramData (hidden)/Solidworks/Solidworks20xx/templates)

The new template should now appear as an option when you make a new drawing.

CHANGE PAPER COLOR - this is a SYSTEM option (Colors > Drawings, Paper Color)

### 9.4.4 - Custom Drawing Sheet Format

Solidworks has its own sheet formats, but you can create your own custom one if you want.

Open a new drawing. Select a SW sheet format that is close to what you want.

Edit the format (RH click in white area > Edit Sheet Format).

Edit the border and title block as desired.

Some title block fields use a simple text block. In that case you fill in the field manually by clicking the text box and typing in what you want.

Other title block fields are "smart" in that they auto-populate. STEPS to do this:

1. Create a text block. Clear it out.
2. On right pane, select "Link to Property"
3. Click "File Properties" - a number of pre-written properties appear. You can create your own too.
4. Once set up, go to Properties, enter property values --> sheet format will update

Sheet Format > Title Block Fields ...

Select the new fields to add to Title Block fields (fields turn blue)

The list on the right pane can be re-ordered so you can tab thru the fields more quickly.  
(you don't have to go to Properties window any more)

Once set up, and then you open a PART using this sheet format. If the part has "meta-data", it will all automatically populate the title block (e.g., material type, etc).

\*\*\*\*\* **XXX** \*\*\*\*\*

## CHAP 10 - XXX

### 10.1 Section 1.1

#### 10.1.1 - Section 1.1

1. Renumbers after 1.1
2. YES!
- 3.

ct\_text1 - Interesting text goes **here**.

ct\_text2

ct\_text3

ct\_text4

ct\_text5

```
// ctz_programg  
// ctz_programg2
```

