MER 101: ENGINEERING GRAPHICS/ AT

HOW TO CREATE A SPRING USING SOLIDWORKS

1) Open SW new part.
2) Select the *Helix/spiral icon. Or from toolbar menu* **(Insert > curve> Helix/spiral)**
3) Select a sketching plane. And place it perpendicular on the screen
4) Sketch a circle with the diameter equal to the mid-diameter of the spring or equal to the outside minus the spring wire diameter, Ex. D=2.0 in.

-If your sketch is complete (black), accept it and a helix/spiral window will open on the left side of the screen.
5) In the defined by box list, select for example Height and revolutions. This requires that you know the height and the number of revolutions of your spring and will use these two parameters to generate the spring.
6) Enter the height (Ex.: 6 in)
7) Enter the number of revolutions (Ex.: 10)
8) For good practice, choose a start angle between 0, 90, 180, 270 degrees. This will place your starting point normal to one of the planes.

Note (New in SW2006): *You can also define a spring with variable pitches. For if you pay attention to most of the springs that exist, you will notice that the last two coils are compressed together, meaning the pitch at both ends is equal to the diameter of the wire.*

9) Check the green check button to complete the creation of the helix/spiral. The completed sketch is the path that you will use to generate the spring.

10) Next select the plane perpendicular to the starting point of your created helix.

11) Select the *sketch button*
12) Sketch a small circle (size: diameter of the spring wire, Ex. .125 in) close to the starting point of the spiral.
13) Use *add relations* to first line up the coordinate system with the center of the small circle, then use *add relations* to coincide the center of the small circle and the starting point of the spiral.
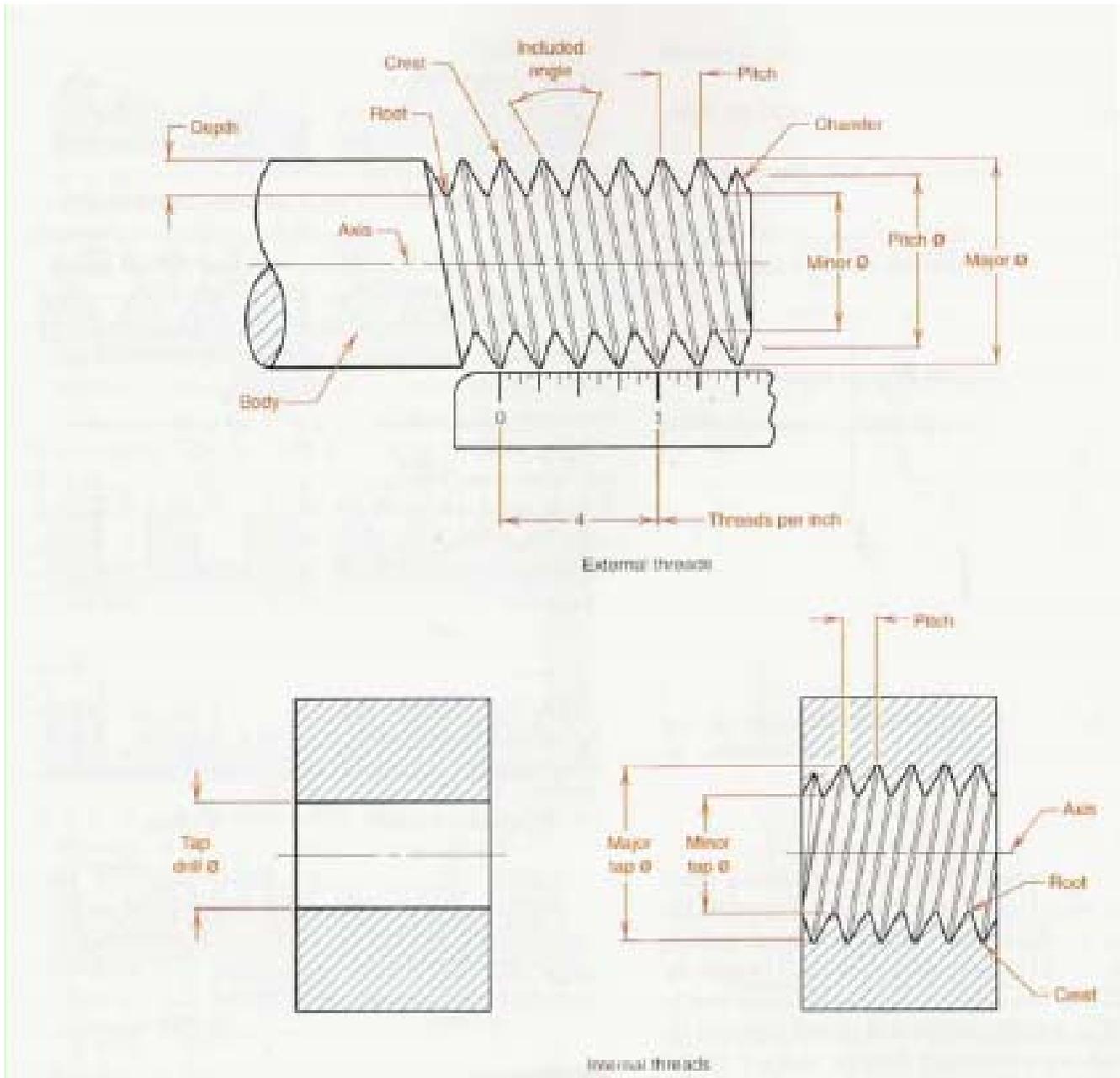14) *Dimension* the small circle (diameter = size of the wire).
15) If the small circle is completely defined (black), exit the sketch and select the *Sweep Icon*.
The circle is still selected. Click in the pink box and select your spiral as the path of the sweep.

16 Et voila ! you have just created a spring.

HOW TO CREATE THREADED PARTS IN SOLIDWORKS

A) INTERNAL THREAD

1) Have pre-created part
2) Select hole Wizard from the Features toolbar
3) Click on the location where you want to place the threaded hole
4) In pop up hole specification window select Tap type

5) Pick the proper standard hole (for example ISO, ANSI inch or ANSI metric)
6) Define the size (for example in ANSI inch Standard ¾ -10 mean ¾ major diameter and 10 threads per inch, and, M 8 x 1.0 means major diameter is 8 mm and the pitch is 1.0 mm in ANSI metric standard).
7) Define the end condition (for example Blind, trough all, Up to next)
8) Options (for example show or do not show cosmetic thread).

B)   EXTERNAL THREAD

Method 1:

- Create the feature using the major diameter.
- Select the edge of the feature.
- Select 'Insert...Annotation.....Cosmetic Thread'.
- Diameter of the cosmetic thread is the root diameter of the threaded part you are created.

Method 2:

In case you need (or want) display the real threaded piece, you will have to create a helical cut or boss using the SWEEP feature. The Sweep feature uses 2 sketches, one as the profile and one for the path. The path is a
helical curve and the profile would be the thread cross section.

1) Start by making the solid rod, (Root diameter if you want to make a sweep boss or major diameter if you want to make sweep cut).

2) Create a helix. (see how to create spring in SW).

3) Create the profile of the thread at the starting point of the helix

4) Select the sweep feature from the toolbar menu and the prompt select the profile and then the helix as the path.

5) Select the appropriate end condition.

COMMON TYPES OF HOLE

Common hole types are shown:
__A. Drilled through.
__B. Blind drilled (not through).
__C. Counterbored hole.
__D. Countersunk hole.
__E. Threaded hole.
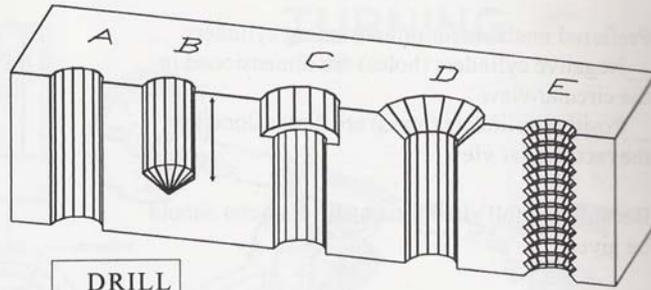
Each of these hole types requires a stand-
ard-form note. Information is given in the
order needed during manufacture. Tools
are sized by diameter, so diameter must be
specified.

Drilled holes are assumed to be drilled
through unless a depth is given.
Drill depth does not include the point.
Drills are available in:
__Fraction sizes. 1/64" to several inches
diameter.
__Number sizes. #1 to #80.
__Letter sizes "A" to "Z".
__Decimal inch sizes.
__Metric sizes.

For counterbored holes specify:
__drill diameter
__counterbore diameter
__counterbore depth
__number of holes (if applicable).

Countersink. Specify:
__drill diameter
__diameter at top of countersunk hole (this
controls the depth).
__angle of countersink (82 or 110 degrees)

Threaded hole. Thru hole.
__tap drill size (from tables)
__thread nominal size
__threads per inch (or mm)
__thread series (if USA)
__class of fit (optional)

DRILL
DEPTH

Ø 3/8 - 3 HOLES
Ø 3/8 (.375) - 3 HOLES
#30(.1285) ALL HOLES MARKED 'D'
'F'(.257) 1-1/4 DEEP - 4 HOLES
Ø.250 3/4 DP.
Ø5 TYP (metric example)

5/32 DRILL - 21/32 C'BORE 1/2 DEEP
Ø 5/32 ⊔ 21/32 ∇ 1/2
(BOTH NOTES ABOVE ARE THE SAME)

Ø 20 ⊔ 32 ∇ 16

Ø 11/32 ∨ 3/4 X 82°
COUNTERSINK NOTE

Ø27/64 - 1/2-13UNC
INTERNAL THREAD NOTE