Parametric Modelling using **Solid**Works®

➤ What is Parametric Modelling and Design Intent?

➤ Planning how to Model/ Build Objects and Products

➤ How to Sketch Properly

➤ How to use SolidWorks

➤ Introductory Points to be Rembered

Parametric Modelling & Design Intent

Solidworks_2003.lnk   Autodesk Inventor 7.lnk   Pro_Engineer 2001.lnk   PTC ProDESKTOP 2001.lnk   Solid_Edge_Assembly.lnk   proewildfire.lnk
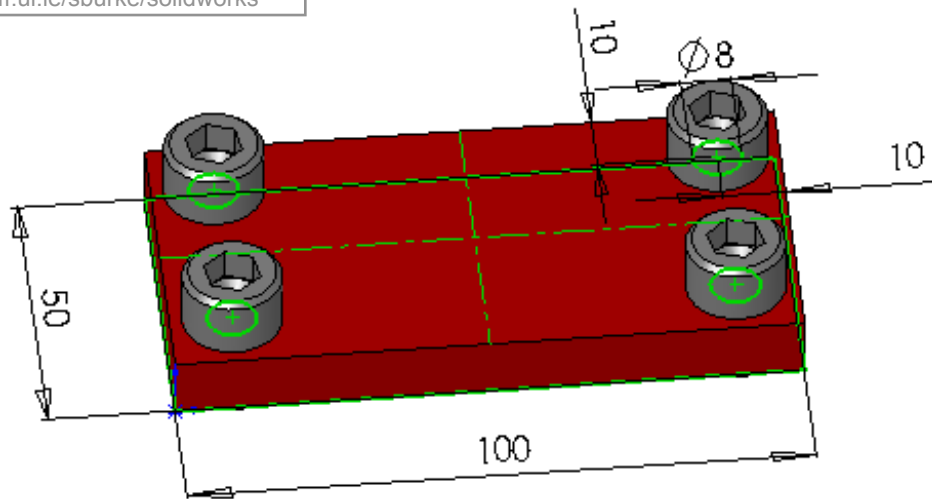
SolidWorks, Autodesk Inventor, Pro/ENGINEER, Pro/DESKTOP, SolidEdge etc. are all Parametric Modelling Software programs sharing common modelling features, techniques and capabilities.

- Learning SolidWorks requires YOU to think in terms of parts and components interacting together, and how these interactions change.

- Components may be holes, chamfers. How can these change within a part?

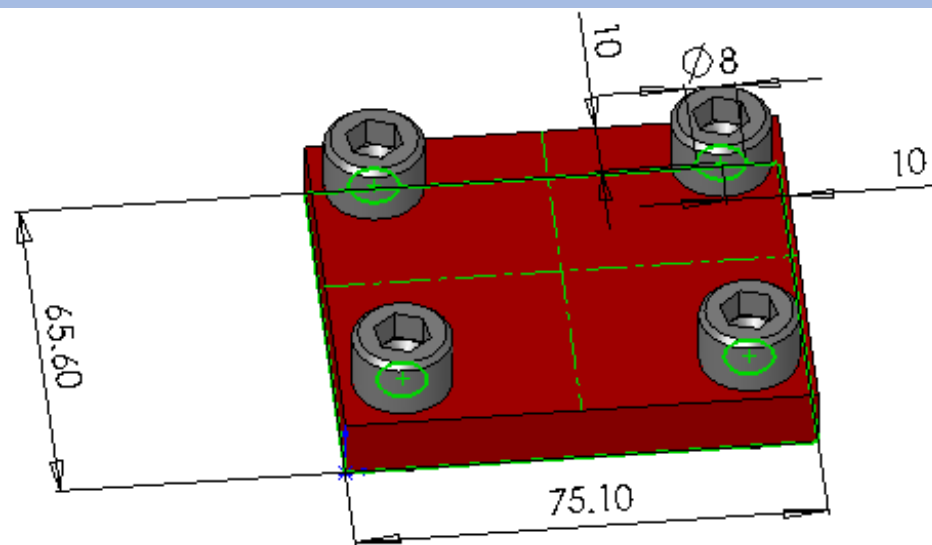- How can parts change with respect to finished products?

The biggest challenge is to think parametrically, i.e.:

"How could this change, and what would change with it?"

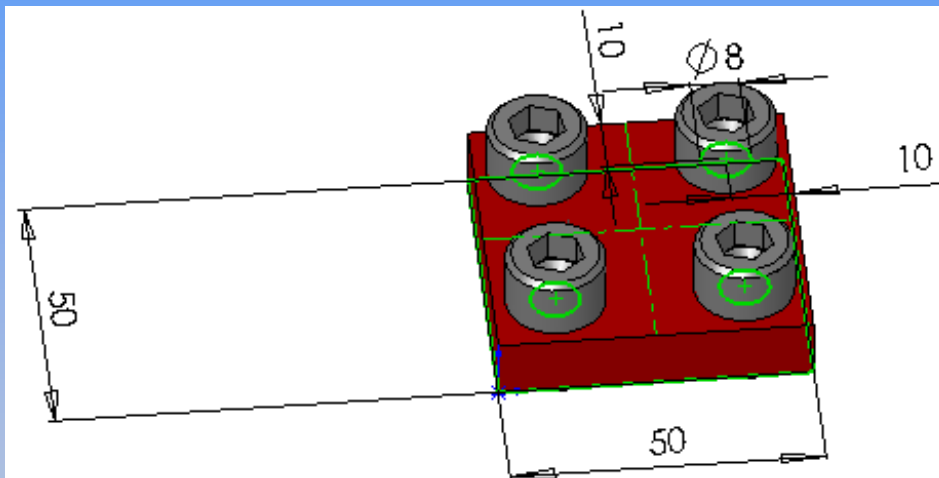and then to create parametric relationships, so everything updates automatically.

Cover Plate
> Correct Parametric Design

The four fasteners are dimensioned in from the edges. No matter what the size of the cover plate, the fasteners will be in the required place.

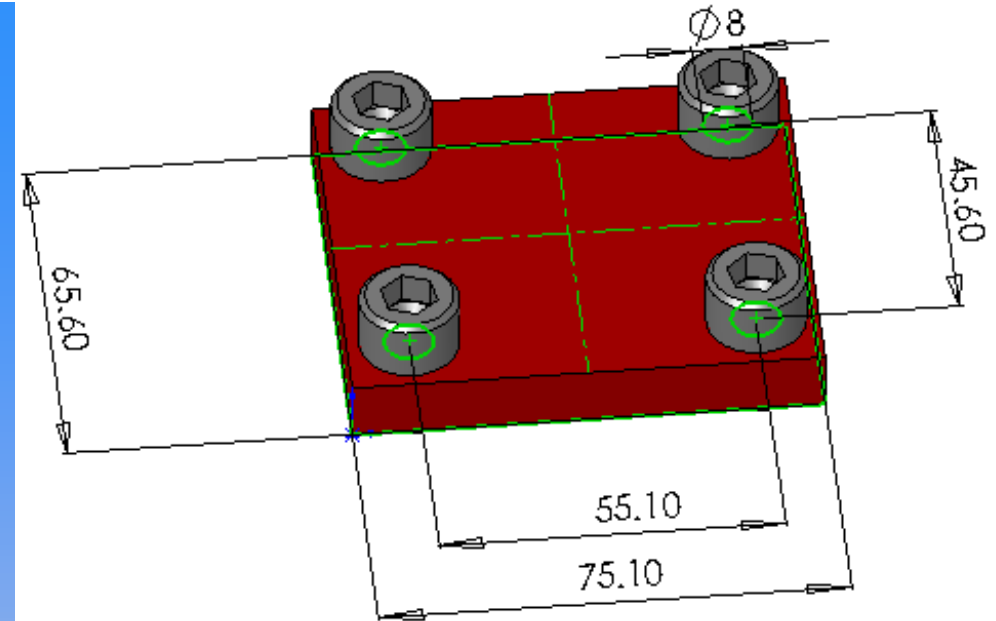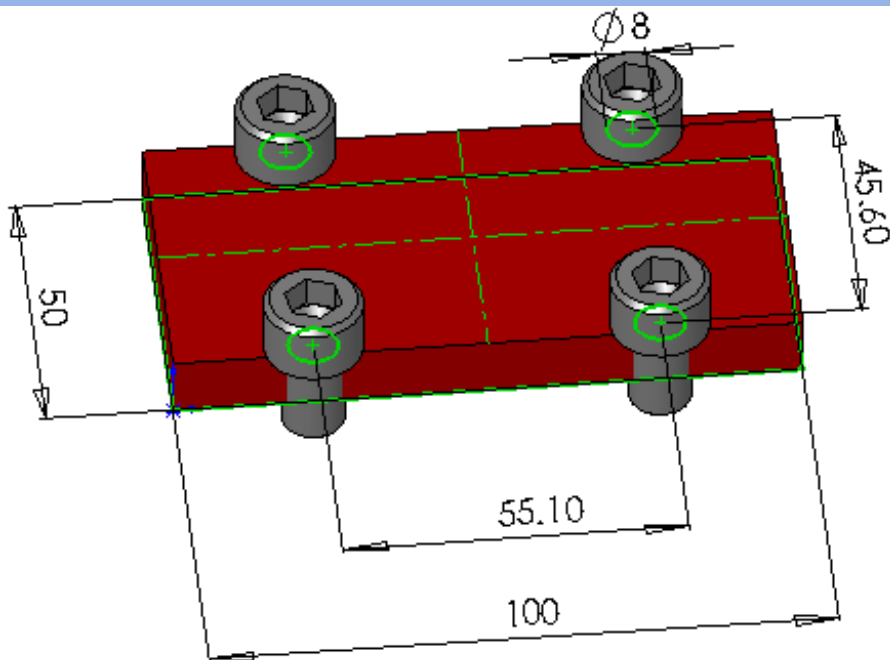Use as much precision as is called for. Do not round dimensions.

Cover Plate

> Incorrect Parametric Design

Result !!

As a Result:

THINK about what is dimensioned to what. Think into the future and how a part may have to change to suit an other part!

Plan how to model objects and products.

- **What goes where?**

  When drawing an object e.g. A Stool, What is drawn first?, where is it drawn? How is a stool manufactured in Industry, in real-life?

  The top of the stool can be designed and produced firstly, then the legs can be made to suit.

  Well we must place the top of the stool on a 'Plane' (work surface) and work from there.

- **What are 'Planes'?**

  Planes are flat surfaces of infinite area. There are three principle 'Planes': the Top, Right and Front Planes. We can draw out the object on one of these flat planes and extrude/ build from there.

- **What Planes to use?**
  (Top, Right, Front)

  What position would the stool rest in real-life? Would the top of the stool be horizontal on the Top 'Plane' or on its side to the Front or to the Right?
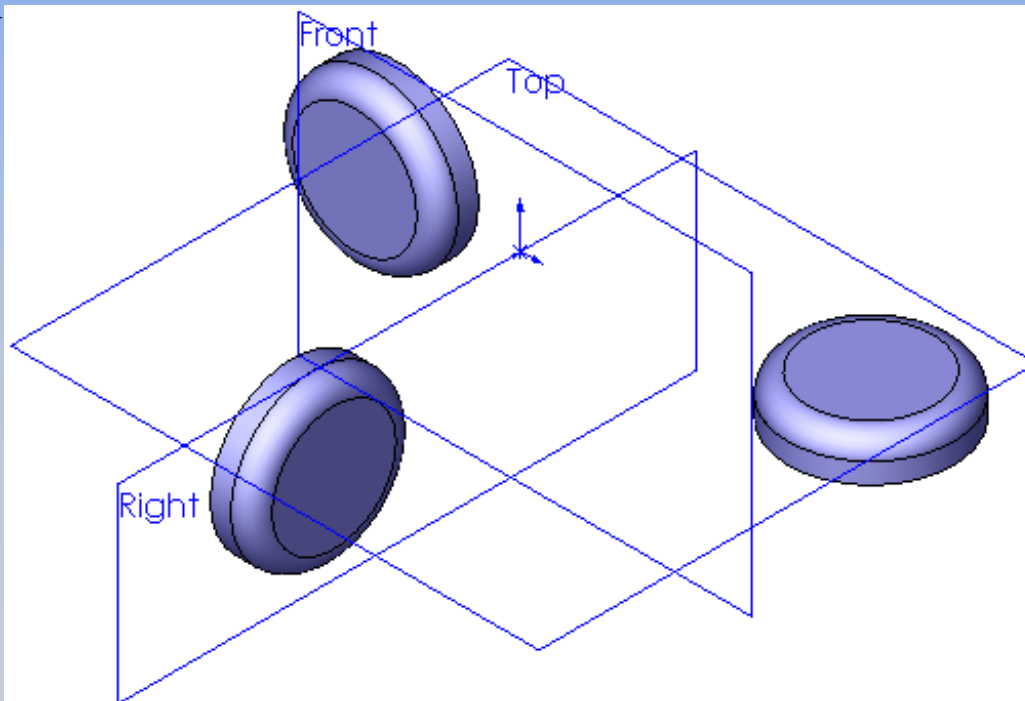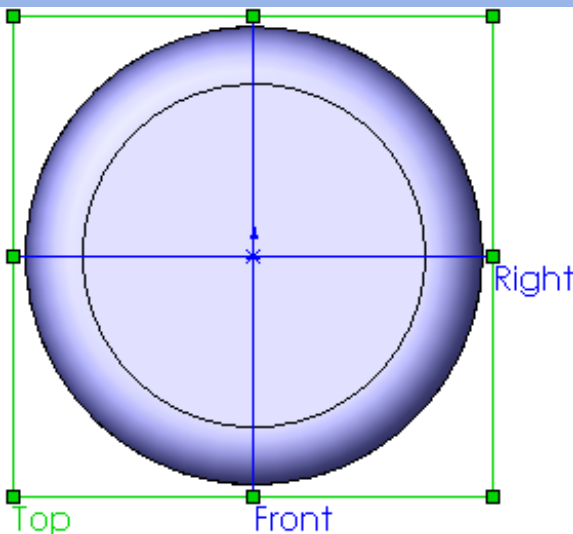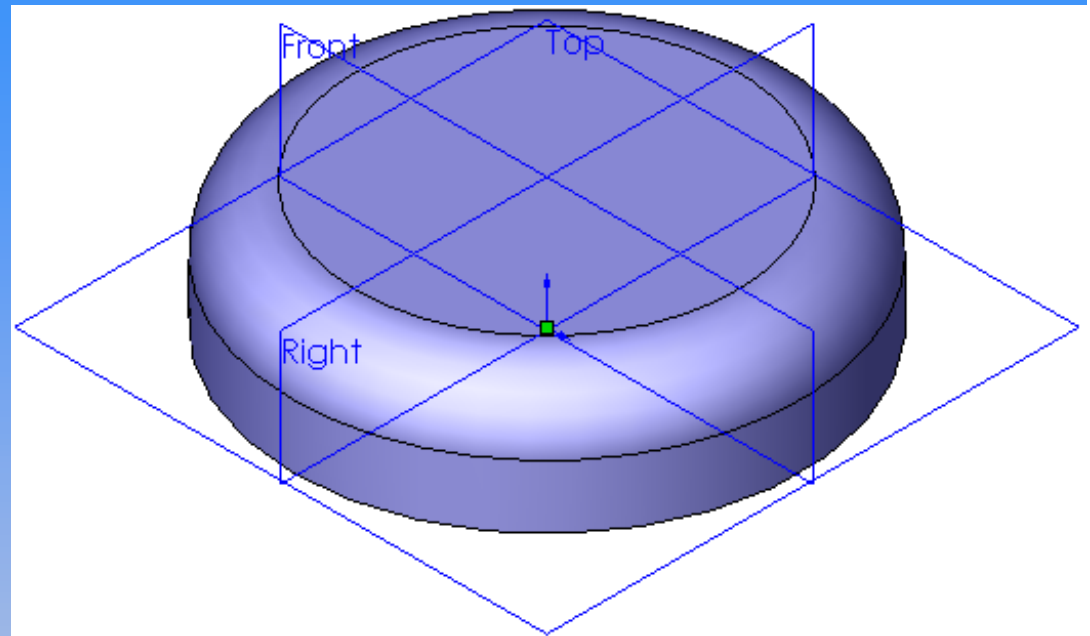
….plan how to model objects and products.

- Answer?

  Correct - The Top Plane!

<u>Take Note:</u>

Objects, especially cylindrical objects are centered on the Plane that they are drawn on, i.e. in the center or *<u>'Centered on the Origin'</u>*.

This is because other Components, (e.g. legs of a stool) and features can be 'concentric' around the origin, and/ or 'symmetrical' between planes. This is not only the case within parts but also can be the KEY to efficient assembling of parts.

….plan how to model objects and products.

- Thus, the legs of the stool can be revolved around the origin or '*central axis*'.
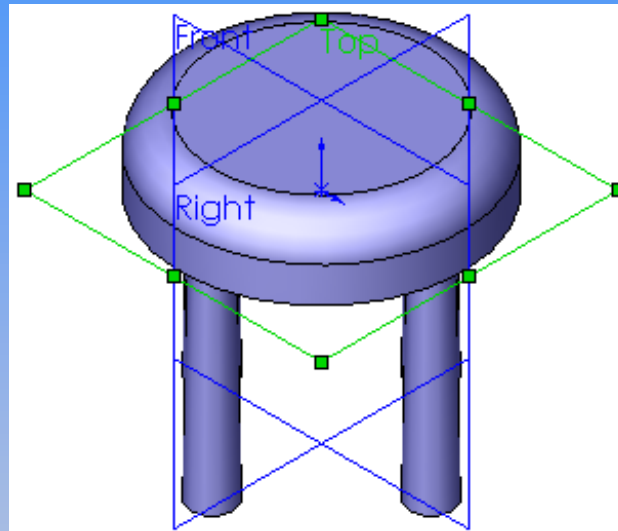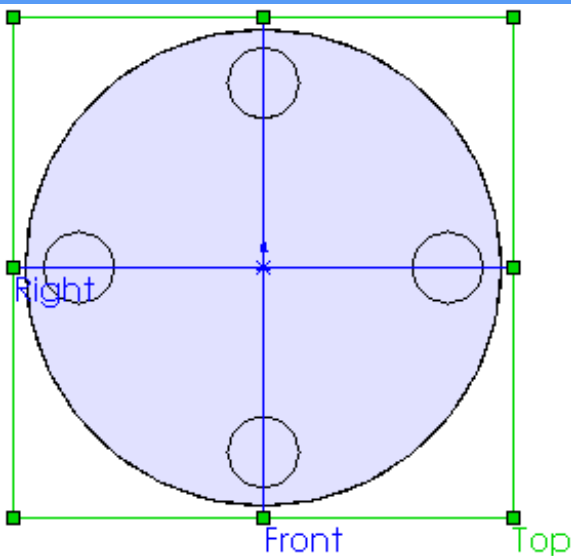
Rendered and Normal viewing as in real-life.

How to Sketch Properly 1.

- Once the correct 'Plane' is chosen, it is of paramount importance that the sketch of the feature and component is

    1. Accurate

    2. Correctly positioned

    3. Dimensioned fully

    - thus having a '_fully defined_' sketch.
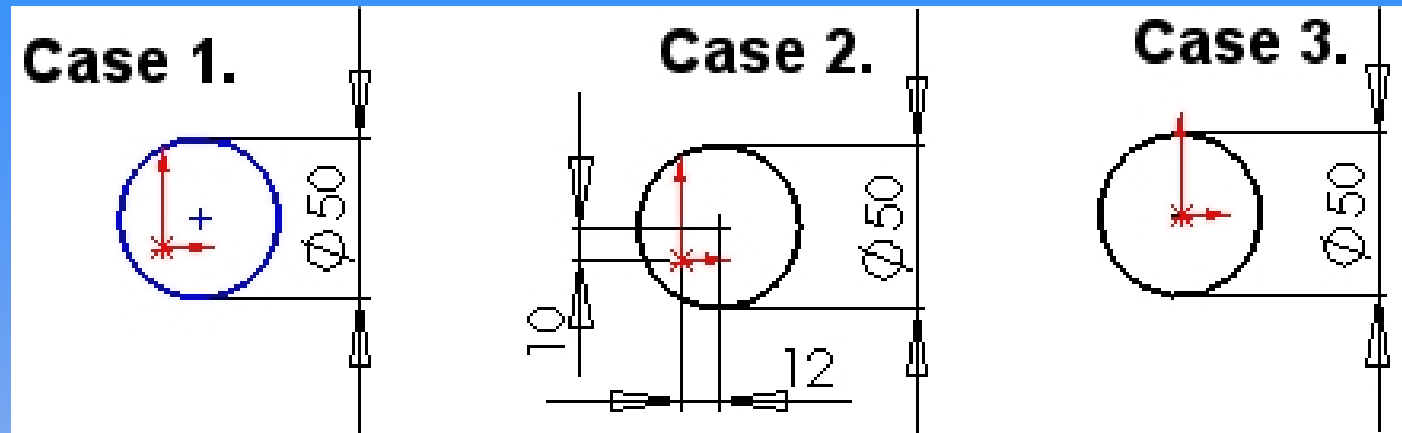
- Why do I have to 'fully define' sketches?

Sketches have to be 'fully defined', otherwise the resulting features and components will be floating, incorrectly related and undefined in space.

Not only do the profiles and outlines have to be fully dimensioned, but they also have to be '_dimensioned from the origin_'.
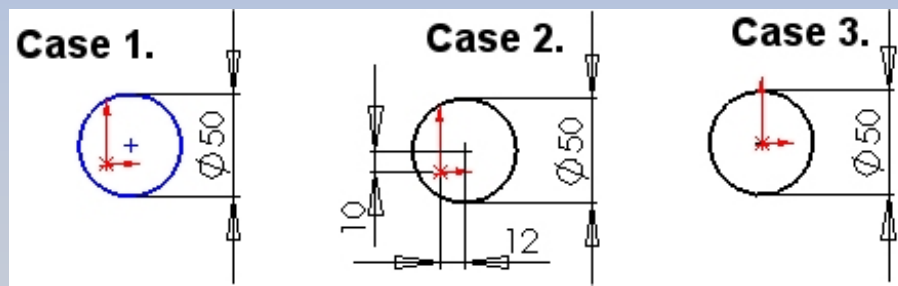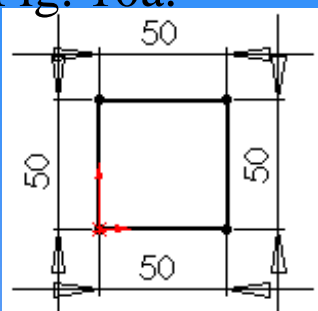
How to Sketch Properly 2.

- <u>Case 1.:</u> Outer profile dimensioned, however geometry still 'blue' and under-defined. Circle can be dragged and moved.

- <u>Case 2.:</u> Fully-defined sketch geometry. Sketch geometry is coloured 'black'. Cannot be dragged or moved.

- <u>Case 3.:</u> Ideal Situation – fully-defined. Sketch geometry is concident with origin.

Fig. 10a.

How to Sketch Properly 3.

Fig. 10b.

Dimensions V's Relations
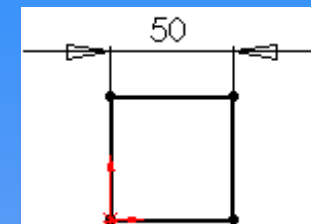
- The IDEAL situation is to use as little dimensioning as possible. Relations such as:

  ❖ HORIZONTAL — Horizontal   ❖ MIDPOINT M Midpoint

  ❖ VERTICAL | Vertical

  ❖ CONCENTRIC ◎ Concentric

  ❖ COINCIDENT ✕ Coincident

  ❖ COLINEAR ⟋ Collinear   …are used to define

  ❖ PERPENDICULAR ⊥ Perpendicular   sketch geometry and

  ❖ PARALLEL ∥ Parallel   allow for parametric

  ❖ EQUAL = Equal   design.

  ❖ MERGE ✕ Merge   Fig. 10a. All sides of square dimensioned

  ❖ TANGENT ⟋ Tangent   Fig. 10b. One side Dimensioned. "Equal"

  ❖ PIERCE ✕ Pierce   relation used on all sides.

How to Sketch Properly 4.
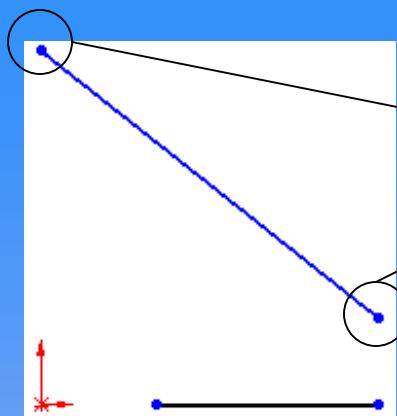
Points V's Lines

- When adding relations, it is important to realise, that a line has a 'Point' at each end. Relations and Dimensions can be used to define either points or profiles in sketch geometry.
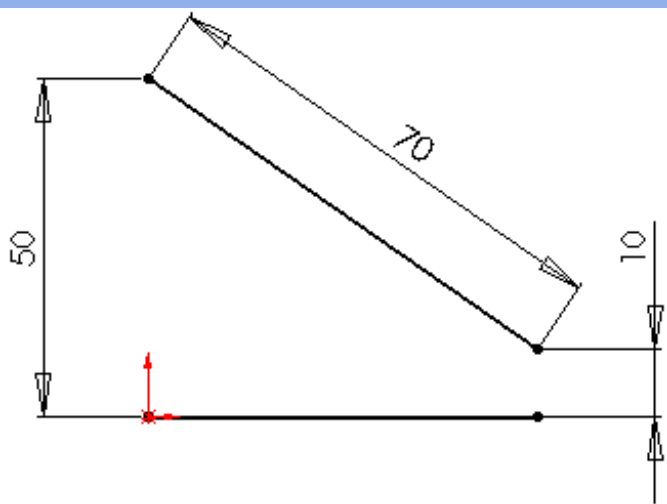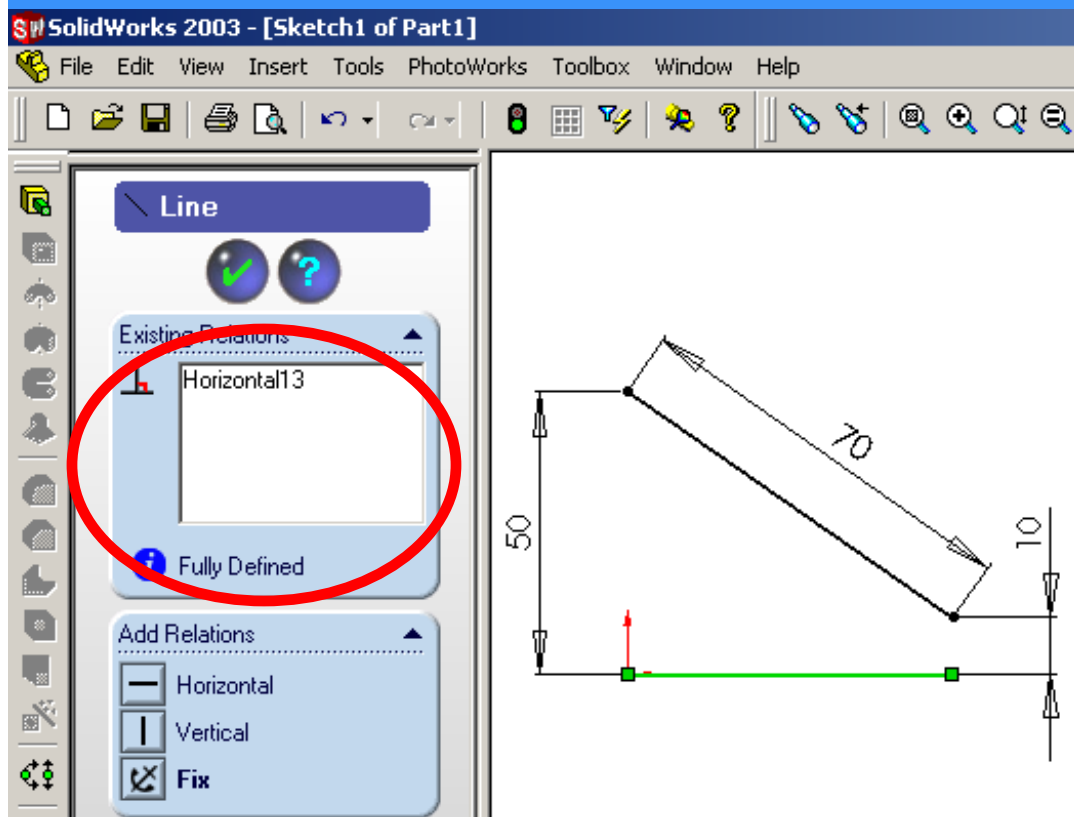
Consider the following incomplete sketch geometry :-

Both dimensions and relations are used defining the points AND the lines in the sketch geometry.

In order to view or change the properties and relations of a line or point, simply click once on the point or line.

How to Sketch Properly 5.

- Properties are shown in the 'Property Manager' on the left of the SolidWorks program.

- We can see, that the line selected (coloured green) has a 'Horizontal' Relation. Nothing more and is Fully Defined.
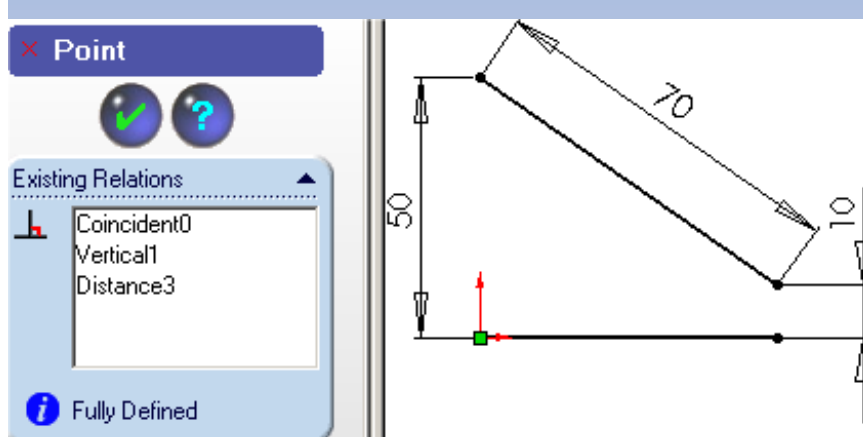
The Selected 'Point' (green) has many relations that you may not have even considered!!!
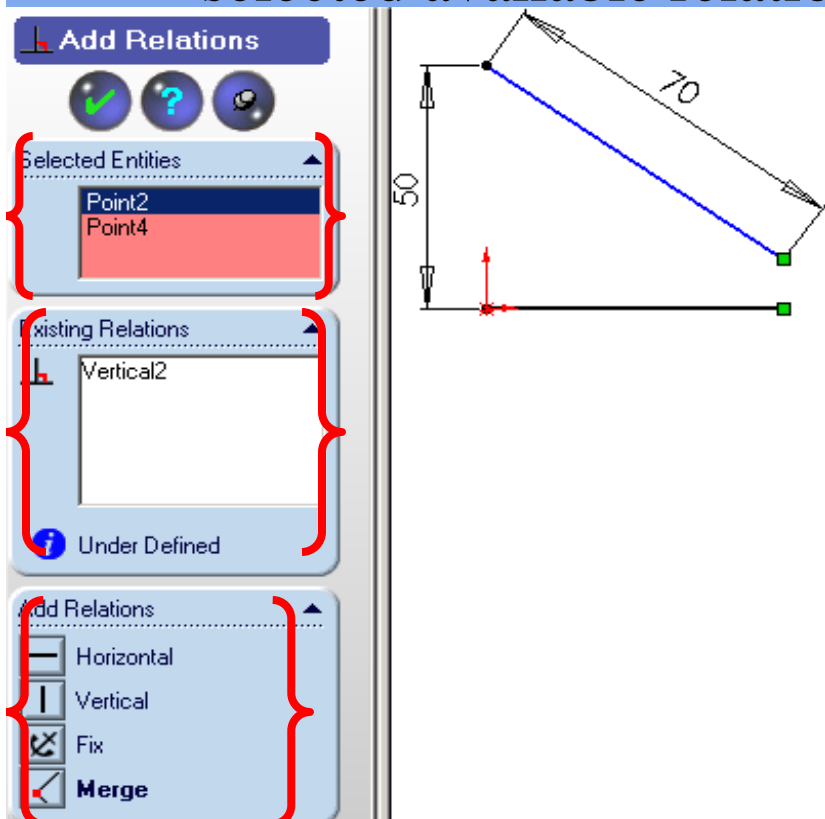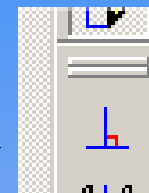
The point is Coincident with the origin.

The point is vertical with the point above it.

The point has a distance (50) relation to the point above it.

How to Sketch Properly 6.
~ Adding Relations

- Relations can be added in two ways:

1. Using the 'Add Relation' toolbar icon.

   Click the icon. The Property Manager appears allowing you select points and profiles. When the appropriate points are selected available relations are shown and can be added.

Add Relations

Selected Entities
- Point2
- Point4

Existing Relations
- Vertical2

Under Defined

Add Relations
- Horizontal
- Vertical
- Fix
- **Merge**

Reading the Property Manager from the Top down,

- The Selected Entities are displayed.

- The Existing Relations are displayed

- The available Relations that can be added in are also displayed.

- The two selected points can be Merged or made Horizontal.

How to Sketch Properly 7.
~ Adding Relations

2.   Using the 'Ctrl' (control) key, click on the appropriate points and profiles, and they can be selected together. Selected Entities, Existing Relations, and Available Relations automatically appear in the Property Manager.

In the Property Manager, similar options

are available for viewing,

adding or deleting relations. If you

want to delete a relation, simply

right click on the 'existing relation'

and select 'delete'.
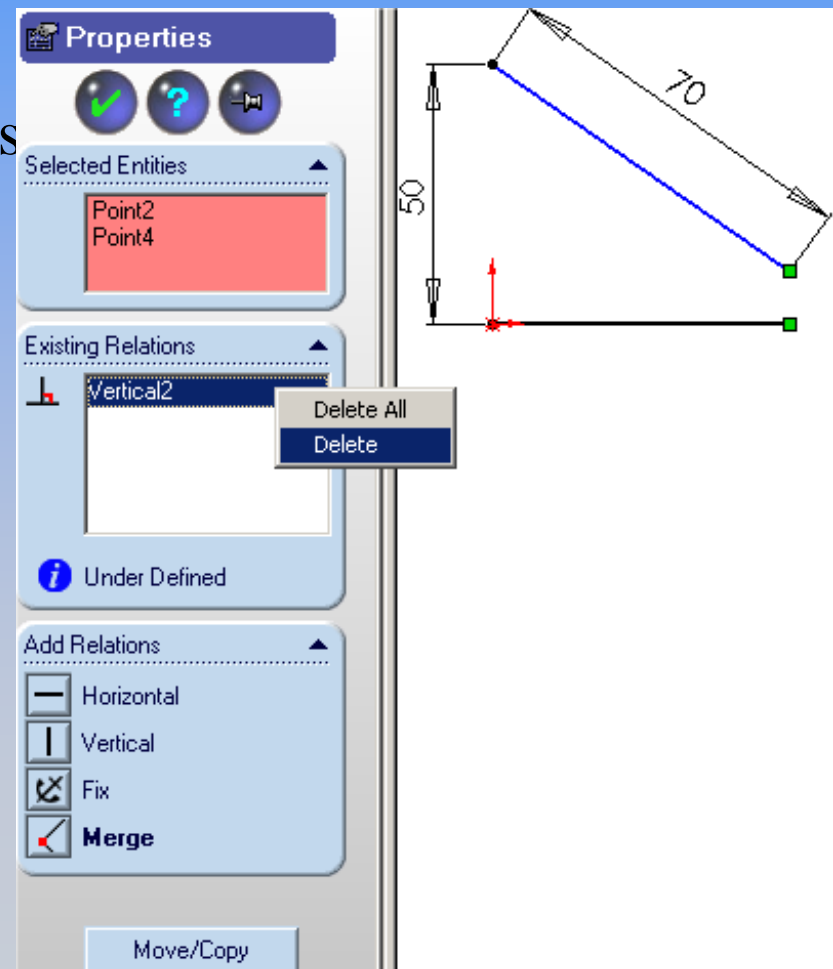
How to Use SolidWorks
~The FeatureManager/ Design Tree
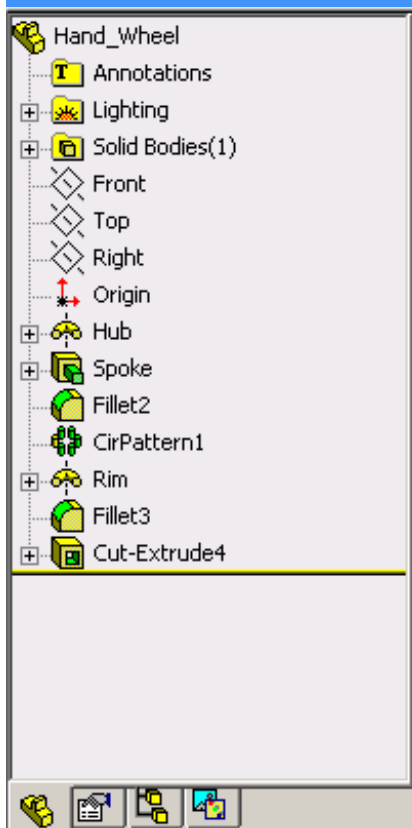
- Everything that is sketched and modelled in SolidWorks is contained within the FeatureManager/ Design Tree. From here properties can be easily and quickly obtained and changed.

- Right Click to get and change Properties

- The ⊞ indicates that there is a sketch 'consumed' and being used by that particular feature. Click on the ⊞ to open and see the underlying sketches.

- Again right click on the sketch to obtain the properties and/ or change them.

- Double click on the sketch or the feature to display the information and dimensions on the graphics area.

Points to be Rembered

"How could this change, and what would change with it?"

- Design correctly using Parametric thinking.

- Choose the correct Sketch Plane (Top, Left, Front)

- Center and Reference from the Origin.

- Fully Define Sketches – Turn Blue Profiles to Black

- Use Relations Rather than Dimensions

- Know how to get and change properties of entities.