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White Paper

Passion Integrity Expertise Teamwork

Robust Part Design Tips

Do your Inventor models fall over at the slightest provocation?

Do colleagues break out in a cold sweat when asked to modify your models?

Do you wade through a sea of error messages when opening a large assembly?

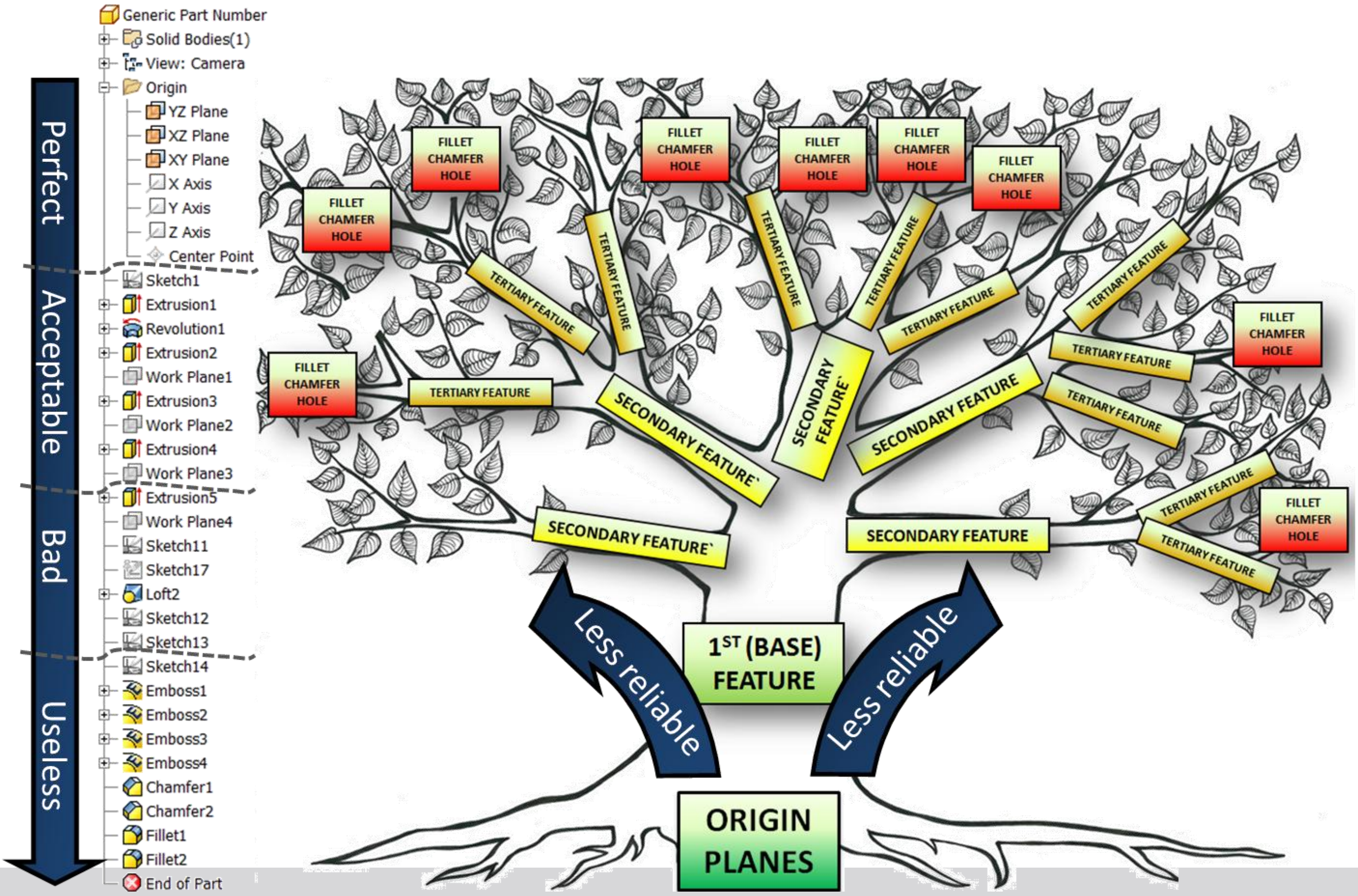
If yes – this document can help you towards making the red cross of doom a thing of the past.

Inventor Users, CAD Managers

The Principle...

Producing robust and reliable models in Inventor is not black magic. Here is the main principle you need to grasp in order to produce parts (and assembles) that are far more reliable:

Use Reliable References! Models Fail When References Fail!



Its called a feature 'TREE' for a reason.

Just like in the woodland variety, **each feature in the tree is dependent on something else (above it, in our case).**

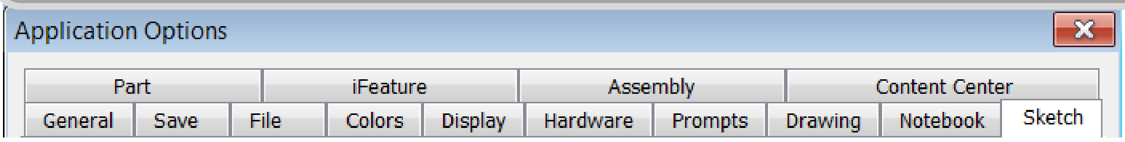
A typical model may have leaves *(fillets and chamfers)* that depend on twigs *(secondary sketched features)* that depend on branches *(primary sketches features)* that depend on the trunk *(first feature)* that depends on the roots *(origin geometry).*

But wait a minute - **You can choose whether to base your feature on a twig or a trunk.**

Obviously you won't have the luxury of sketching every feature on an origin plane, but in 90% of cases you can **sketch on a user plane** directly based off the origin plane (preferably with a named parameter). **This is far more reliable than sketching on a model face as it is closer to the sturdy roots of the model.**

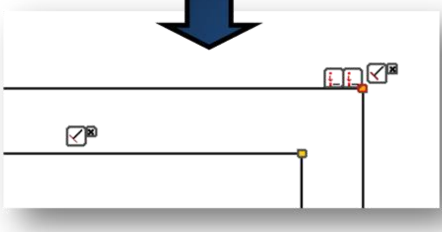
1) Place your sketches on work or origin planes in preference over model faces
2) Make sure that **projected lines** in sketches are **never based on flimsy twigs** in the model tree. (and remember using parameters is FAR more reliable than projecting lines in sketches – more on this later)

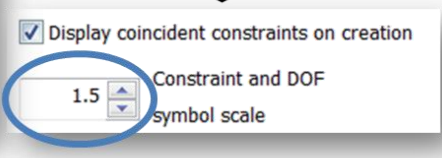
Set Your Application Options…

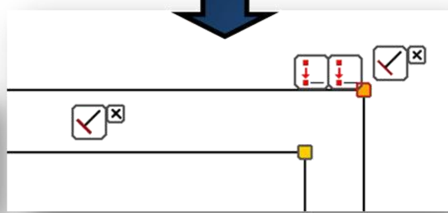


1) Set Your Sketch Options…

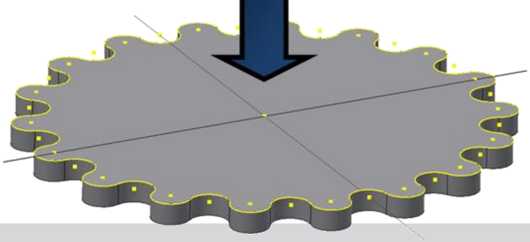
Make your sketch constraints display larger (now there's no excuse for shoddy sketches)…..

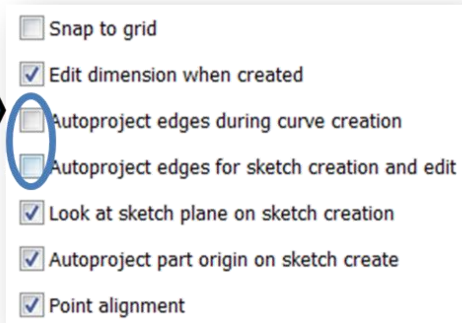


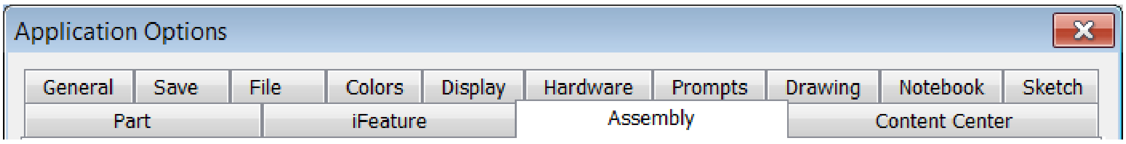




Prevent unwanted lines in sketches (like this recipe for future disaster)….

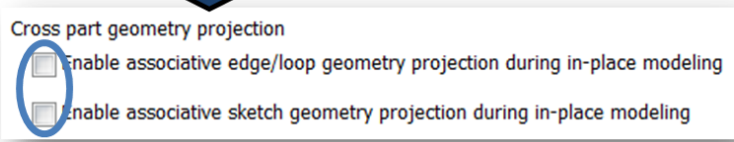






2) Set Your Assembly Options...

Consider 'disabling' associative projection between parts in an assembly (as shown below)........



If these boxes are ticked you may accidentally project linked geometry from other assembly parts while sketching. This can cause problems later when changes are made. Unticking these will only allow unlinked lines to be projected. This is only a suggestion – see tip 'Avoid Adaptivity' below for additional info.

Sketching – General Principles...

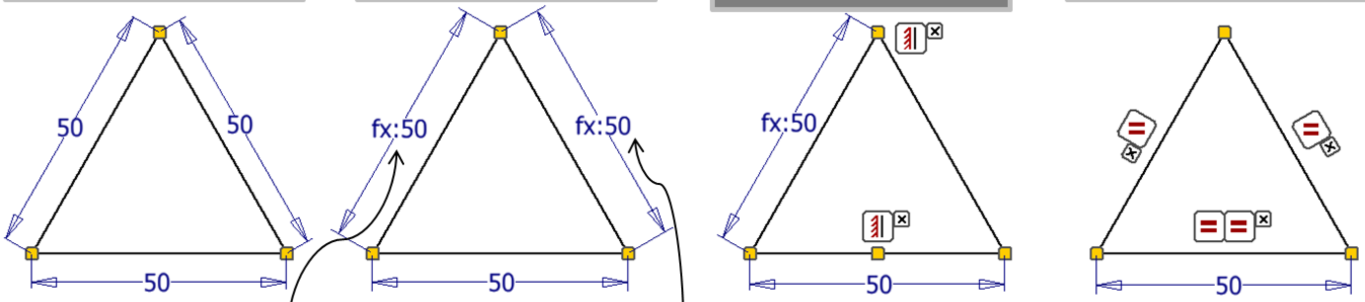
Constraints are better than dimensions...

If you can use a constraint instead of a dimension, DO IT.

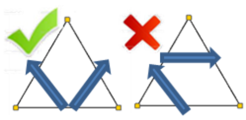
1) Your sketches will be less busy.
2) Your design intent will be less likely to be accidentally overwritten.

4 different methods for drawing an equilateral triangle – which is best?

3 Separate Dimensions RUBBISH	1 Controlling Dimension 2 Referenced Dimensions GOOD	1 Controlling Dimension 1 Referenced Dimension 1 Vertical Constraint PERHAPS	1 Controlling Dimension 2 Equals Constraints BEST



One of these could be overwritten by a novice - removing your design intent



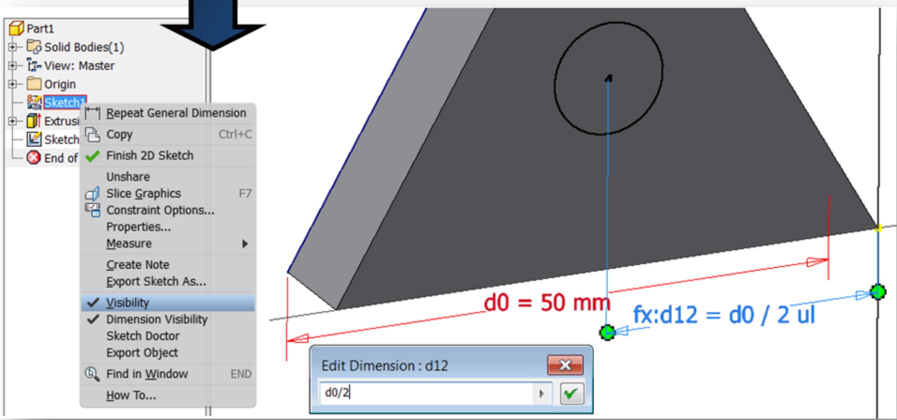
Reference both of these dimensions to the controlling dimension not to each other!

Thanks to Sean Dotson for the principle here...

Linked dimensions are better than projected geometry……

How do I do this?

How?

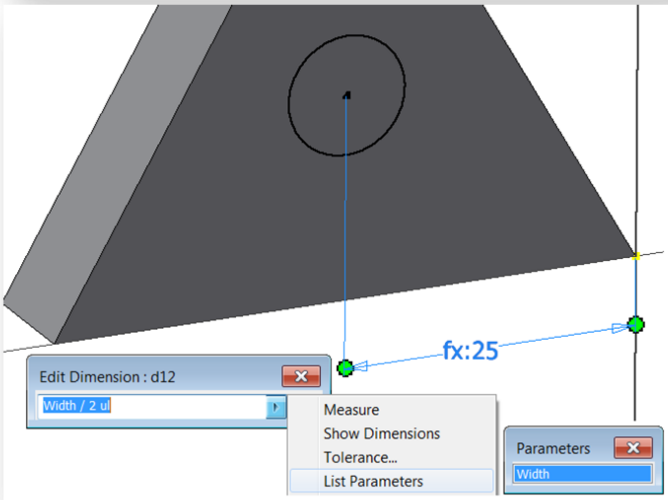


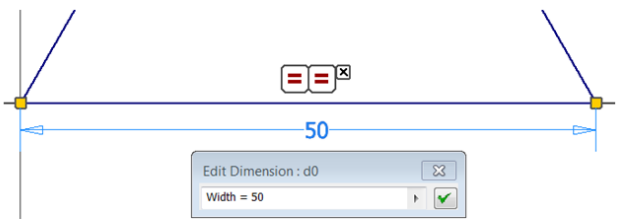
Method A - Select the dimension graphically

1) Make a previous sketch visible.

(note this can be done in an active sketch)

2) Click on a dimension in that sketch to reference it.



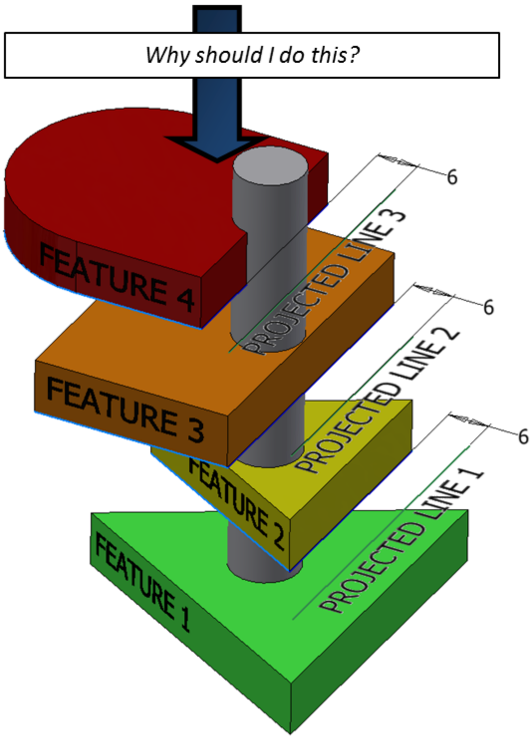


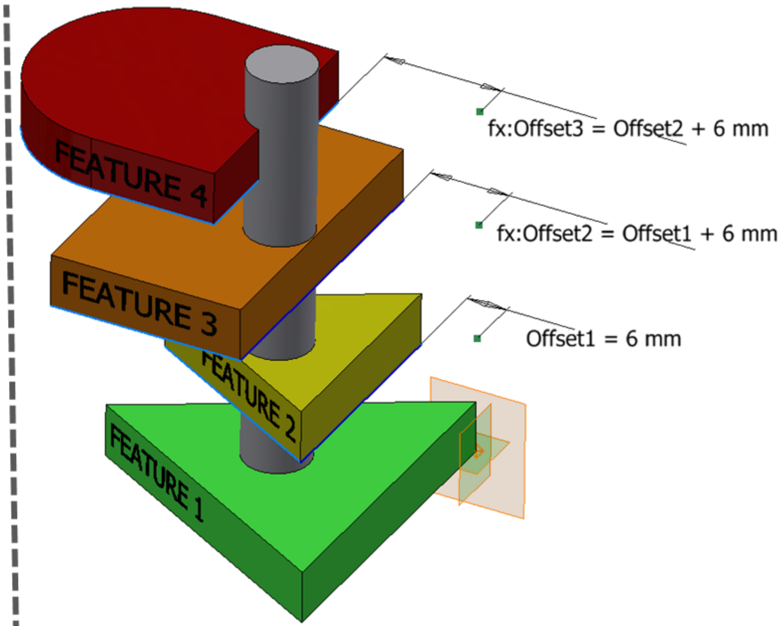
Method B – Select the parameter from a list

1) When placing dimensions in a sketch, type in a name for the parameter (e.g. Width = 50 shown above)

2) Any named parameters can then be referenced using the 'list parameters' option (shown left)

Why?





What happens here if feature 1 changes?

Inventor must attempt to update a chain of 3 projected lines in order to update feature 4. Any link in this chain can break.

Long chain of dependency =
Building on TWIGS! =

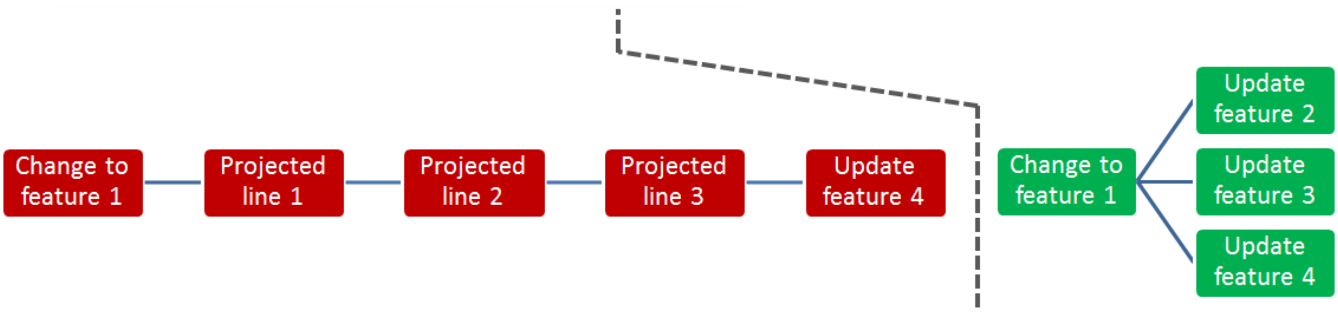
BAD!

What happens here if feature 1 changes?

The offset of features 2-4 are controlled only by linked parameters and the (projected) origin point. Therefore by updating 1 parameter Inventor can update position of all the features.

Short chain of dependency =
Building on ROOTS =

GOOD!



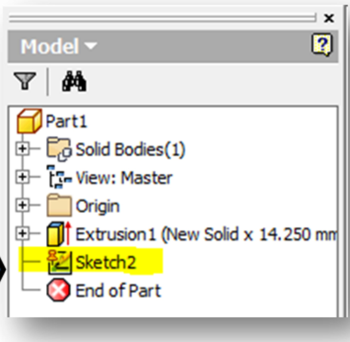
Sketching – Tips for Robustness...

Tip: Fully constrain all sketches!

Ignore this and you may as well be modelling with jelly!

This is so important that there is a special icon for a fully constrained sketch (see picture)

Return any models containing unconstrained sketches to their maker and demand satisfaction!



Tip: Avoid Adaptivity.

A part will become 'Adaptive' in an assembly when a feature of another part (in that assembly) is projected as lines into a sketch within that part. This means it will (attempt to) update if the part it refers to changes.

Here's what an adaptive sketch looks like: 

Either:

1) *Disable it with application options **(see section above for details)** – OR........*
2) *Hold CTRL key when projecting to break the link. OR........*
3) *Right click to switch off adaptivity when you've finished referring to other parts in the assembly. OR........*
4) *Use with extreme caution. Accidental adaptive parts are one of the biggest threats to robust modelling in Inventor.*

Tip: Don't put fillets and chamfers in sketches.....

*Remember the Model Tree of Dependency – **fillets do not make good foundations for other features.***

As a general rule, placed features such as fillets and chamfers should be LAST in the model.

(There are exceptions – such as if you need to fillet before shelling a part for example).

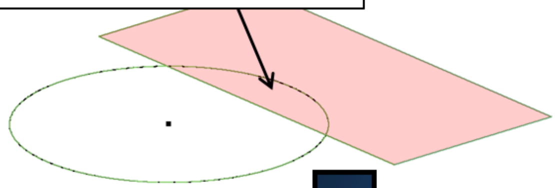
Tip: Don't use more than 10-12 lines in a single sketch.....

Inventor slows down with too many sketch entities,
Minimising the number of sketch lines also reduces the chances of user error when constraining

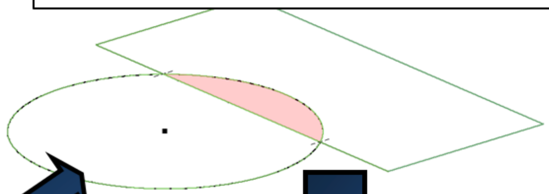
Tip: Don't draw lines on top of other lines.....

Drawing duplicate lines to allow you to extrude or revolve a profile is bad practice and may lead to errors later.

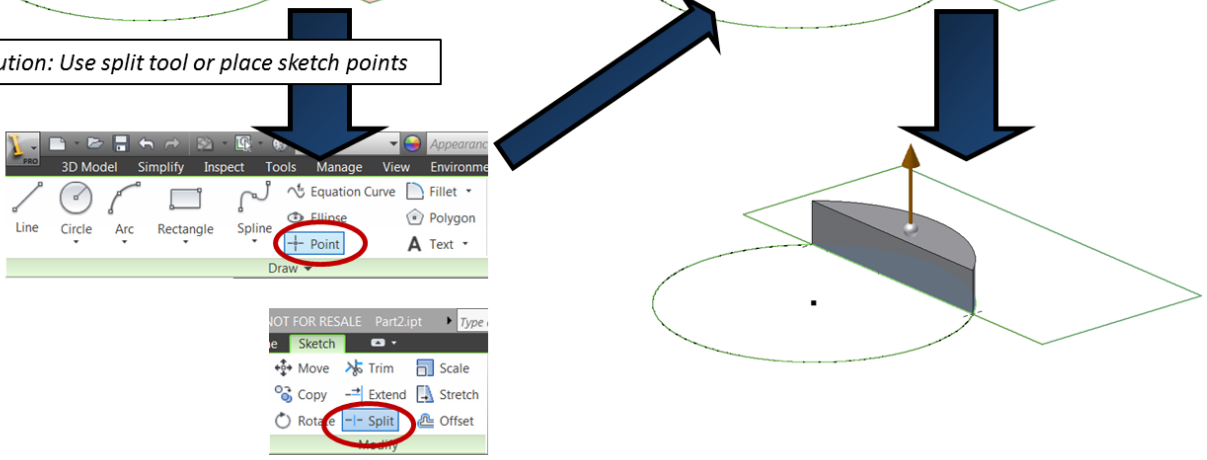
Can't extrude an intersection area?



Now intersected area can be selected as a profile



Solution: Use split tool or place sketch points

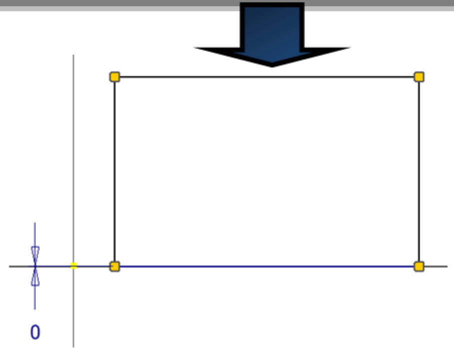


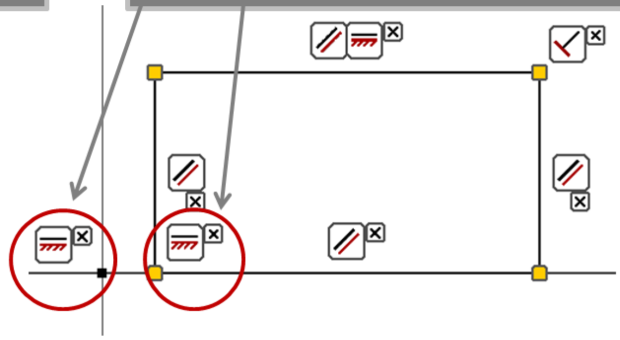
Tip: Don't use zero dimensions in sketches

***Case 1:** If its always going to be zero, then use a constraint instead.*

This clutters up your sketch and may allow unwanted rotation

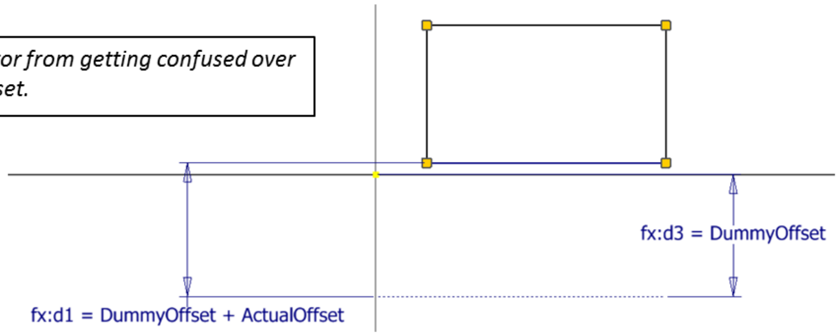
This will lock the sketch down correctly (select the two points you want to be aligned)





***Case 2:** If it may vary <u>across</u> zero (with possible negative offsets also), then use an offset reference line.*

This will prevent Inventor from getting confused over the direction of the offset.

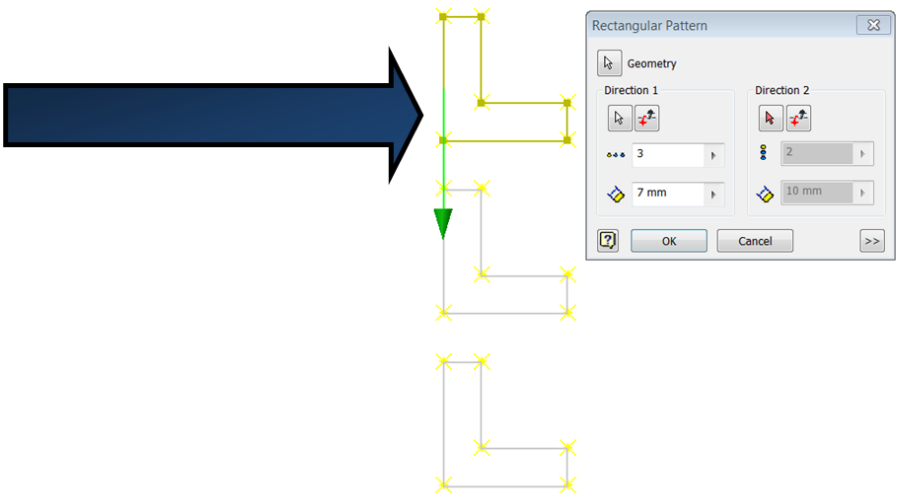


Here - 'ActualOffset' is a user parameter that can be positive or negative – without negative consequences ☺

Tip: Don't use patterns in sketches.....

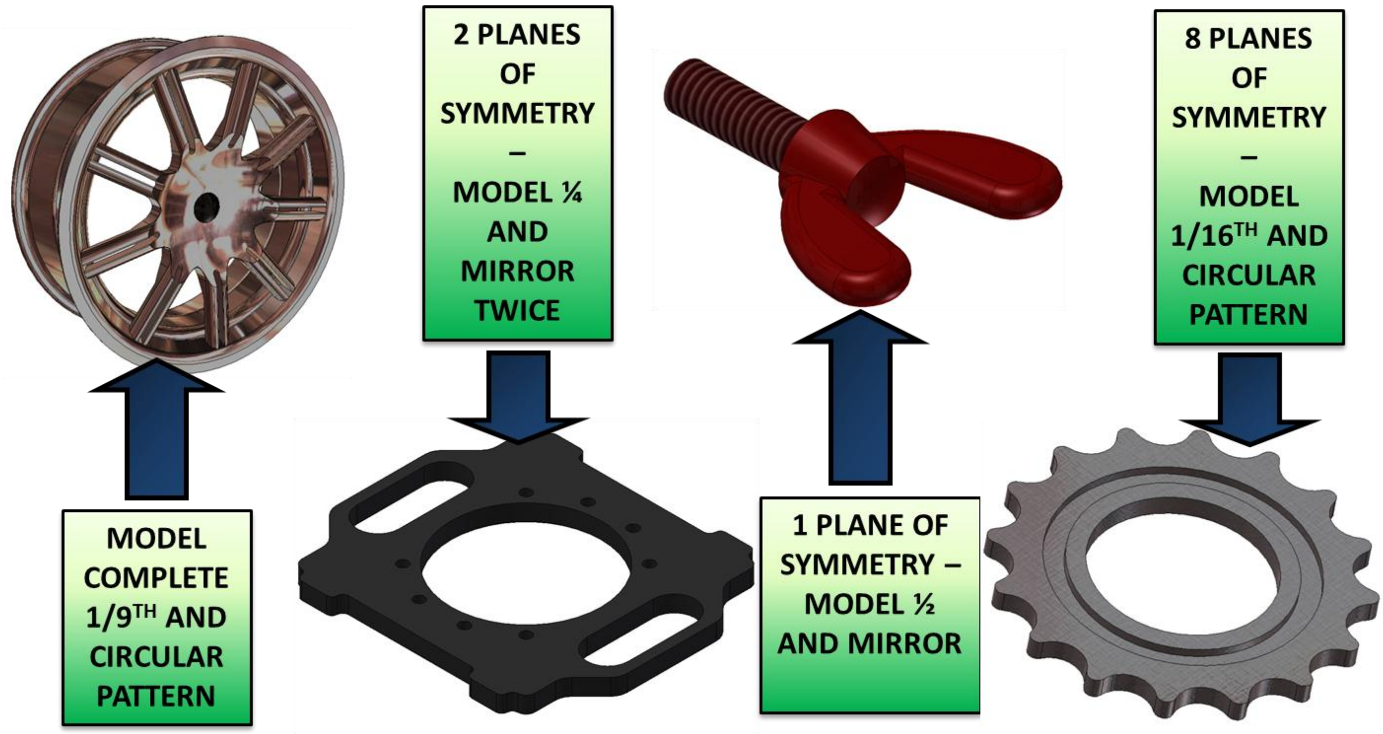
Why?

1) **It'll slow the sketch down.**
2) **It's fiddly to edit.**
3) **If the quantity of the pattern changes (from 3 to 5) then the feature based on this sketch won't update (I'll still have 3 'L' features).**



Modelling – Tips for Robustness…

Tip: Take advantage of planes of symmetry and patterns……



Tip: Consider using the ‘Shell’ command – very powerful when combined with mirroring etc……