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WORKSHOP FILES

Please download and extract files from the following link:

<http://go.unimelb.edu.au/p33n>

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 - [@resplat](https://twitter.com/resplat) or [@bobbyli22](https://twitter.com/bobbyli22)
- Come to our #hackyhour for consultation, or if you just want to have a talk to us over some beers.
 - Every Thursday at 3:00pm – 4:00pm
 - Located at Tsubu Bar



 Katie Ewing
@katieewing

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we even got an umbrella at #hackyhour this week! @ITS_Res

3:33 PM - 13 Mar 2014

4 RETWEETS 1 FAVORITE



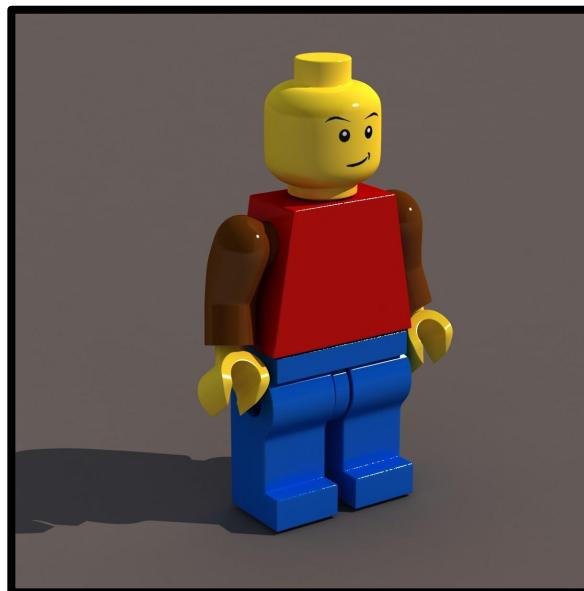
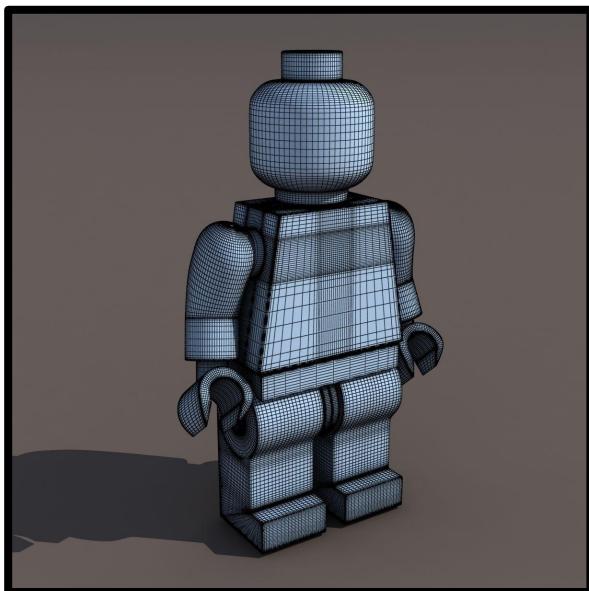


INTRODUCTION TO COMPUTER-AIDED DESIGN WITH AUTODESK INVENTOR

DESIGNED FOR RESEARCHERS

WHY USE COMPUTER-AIDED DESIGN (CAD)?

- Create 3D Models, Assemblies, and Simulation
- Product Realisation (e.g. 3D Printing)



BASIC MODULES OF 3D DESIGN

■ PARTS:

- Sketch a 2D profile
- Extrude a 3D solid based on the shape of 2D profile
- Add details to 3D solid

■ ASSEMBLY:

- Import different PARTS to form a design

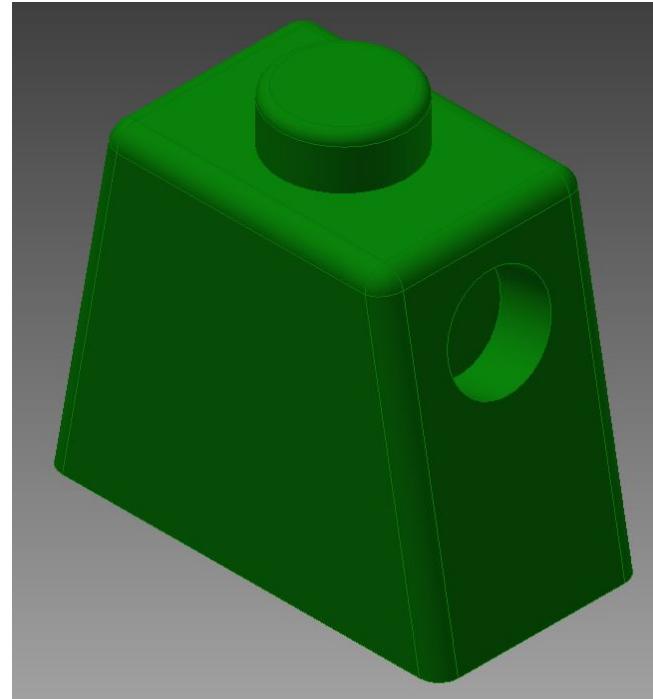
■ PRESENTATION

■ DRAWINGS:

- Create precision drawings to represent PARTS and ASSEMBLIES

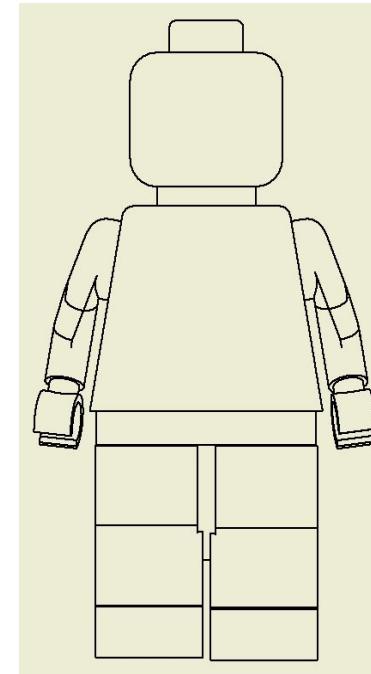
WHAT ARE YOU LEARNING?

- Introduction: How to create a PART and EXPORT for 3D Printing



INTERESTED IN LEARNING MORE?

- Advanced: How to connect PARTS in an ASSEMBLY, and create technical DRAWINGS



LESSON PLAN

- 9:30am Introduction
- 9:40am Lesson 1: Interface & Navigation
- 10:00am Lesson 2: Basic 2D Sketching
- 11:00am Lesson 3: Dimensions & Constraints
- 12:30pm LUNCH TIME!
- 1:30pm Lesson 4: Basic 3D Modelling
- 3:00pm Lesson 5: Modelling Example, Visualisation and STL Conversion



INTERFACE & NAVIGATION

LESSON 1



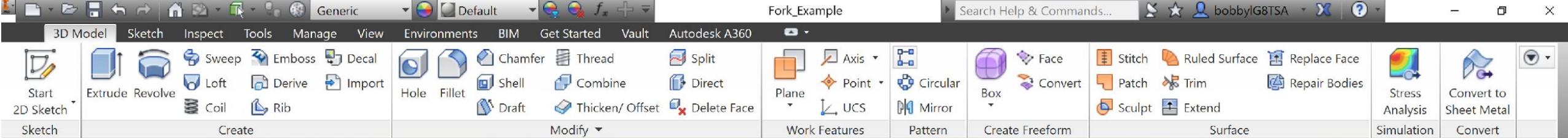
GETTING STARTED

- Let's jump straight into using the program. *Open your inventor program.*
- Before we start making new designs, let's look at an existing file to get familiar with the interface.
- Open an existing file: **Fork_Example**
- *The file can be found in the “O1_Part” folder in the downloaded materials.*



THE INTERFACE

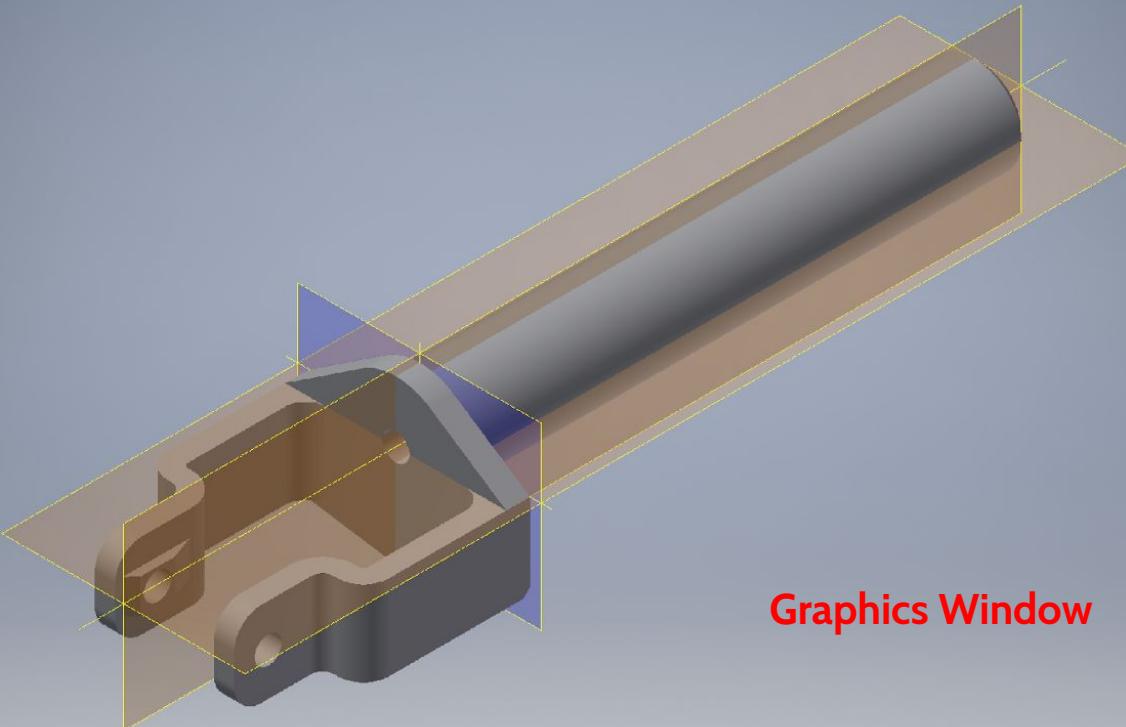
- The style of the Inventor could be quite familiar to you, but if not hopefully it won't be too hard to pick up:
 - All command options are located in the top **ribbon** under different tabs.
 - The **graphics window** is your workspace where your designs will appear.
 - There is a **navigation cube** that will determine the orientation of your view of your model.
 - The **model browser** is the history of your model, and will contain all information about it.
 - In the bottom left-hand corner there is a **command status** that will prompt you whilst you are designing, if you're not doing anything it will just say "ready".



Model Browser



Command Ribbon



Graphics Window

Command Status



My Home Fork_Example.ipt

Navigation Cube



Ready

Ask me anything



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NAVIGATION CUBE

- You can find your navigation cube in the top right-hand corner, and its your little buddy when it comes to orientating your model.
- The cube corresponds to the xyz coordinate system in the bottom-left corner of the graphics window.
- Selecting any element on the cube (face, edge, corner) will reorient the model. *The model is fixed to the xyz coordinate system so you will see that it also rotates.*
- Selecting the **home** icon will return the model to its original orientation.
- Any preferred orientation can be set as the home view:
 - Right-click home icon
 - Set current view as home

NAVIGATION COMMANDS

- There are three primary modes of movement in the 3D environment: ***translation, rotation and scale (zooming in and out)***.
- You can select any of these commands from the navigation wheel found under the **view tab** or on your **navigation menu** on the right of your screen.
- But, you can do these much more efficiently by using the keyboard bindings or using a three-pointer mouse (which you should have with you!)

NAVIGATION COMMANDS CONT.

- **Translation:**
 - Hold **F2**, then hold your left mouse key and drag to move around.
 - Hold **middle mouse wheel** and drag to move around.
- **Rotation:**
 - Hold **F4** to make the rotation wheel appear, then hold your left mouse key and drag to rotate.
 - Hold **shift + middle mouse wheel** and drag to rotate.
- **Scale (Zoom):**
 - Hold **F3**, then hold your left mouse key and drag to zoom in and out.
 - Scroll the **middle mouse wheel** to zoom in and out.
- *If your computer is anything like mine though, you will have to hold the “fn” key to use any of the function hotkeys which can get annoying so try to use your mouse as much as you can.*

MODEL BROWSER

- The model browser is a history of the how the model was created in chronological order.
- Each item is a component in the 3D model and they can be moved around, but some are related and moving them cannot be possible due to their dependencies, *i.e. primary and secondary 3D features, but more on this later.*
- If you expand any primary component in the model browser, you might find there is a 2D sketch underneath it. *There is a relationship between the 2D and 3D elements of modelling.*
- Any of these components, even their sketches, can be edited. *To open the edit menu, right-click on the browser entry.*

EDITING FROM THE MODEL BROWSER

- 1. Open the Fork_Example model.
- 2. Change the diameter of the fork handle from 28mm to 15mm.
 - Edit the sketch associated with the 3D component.
- 3. Suppress “Fillet2”.
- 4. Delete “Hole1”.
- 5. Use “End of Part” to suppress features.

ORIGIN FEATURES

- In the model browser there is an “Origin” folder containing all of the origin work features.
- Work features act as reference entities that assist the designer identify their orientation in the 3D work space. And correspond to the xyz axis of the environment.
- The visibility of these work features can be turned on and off as desired. *Right-click to access the option.*
- Work planes are the most useful in 3D design, they have a positive and negative orientation:
 - An **orange** colour indicates that you are looking at it from the positive direction.
 - A **blue** colour indicates that you are looking at it from the negative direction.

CONCLUSION

- We looked at the basic interface of the **parts** module. Other modules will have different command load-outs because they are used differently, but will have the same style of interface.
- Getting familiar with the navigation commands. **Use a mouse people!**
- The model browser is a record of your design. Use it to find components and edit features to change or optimise your design.
- Work features will come up more often when we start to modelling.



BASIC 2D SKETCHING

LESSON 2

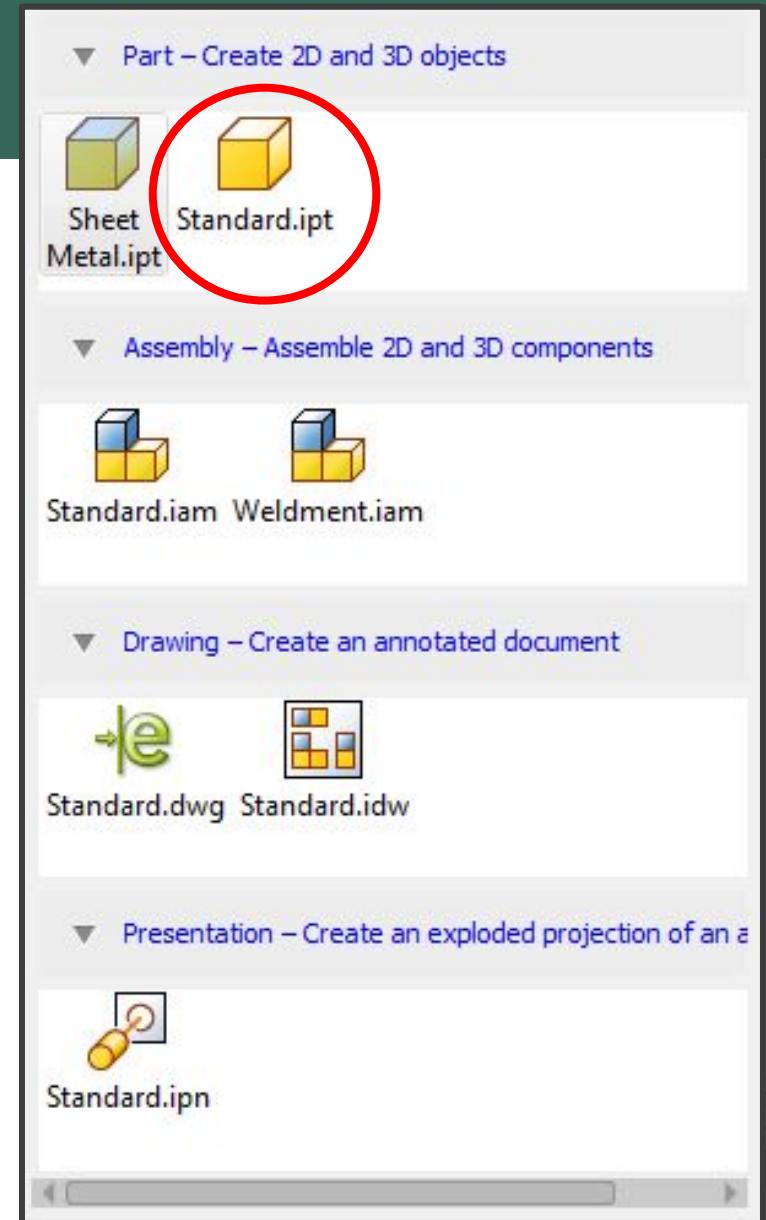


MOTIVATION

- A part file is the most basic fundamental component of a design, but even these parts are made of design features.
- In the **Fork_Example**, we found sketches hidden underneath 3D components indicating that there is a **parent-child relationship** between these elements.
- Therefore it is fundamentally important that we first learn how to sketch the 2D elements, also known as the **profile**.

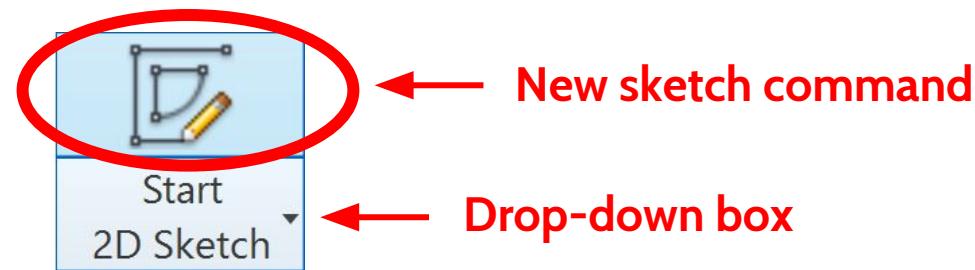
CREATING A PART FILE

- A **.ipt** extension indicates a part file.
- The most basic type of part file is the **standard**, we will be using this throughout the rest of the workshop so always select this one when starting a new file.
- Most people tend to work in the **mm** units, but **inches** is also an available option.
- **Sheet Metal .ipt** files correspond to sheet metal types of components and will not be covered in this course.



CREATING A PROFILE

- Let's begin our sketch with the **new sketch** command. *Click on the box above not the drop-down tab, if you do accidentally get the wrong command make sure that the sketch command is set to 2d sketch.*



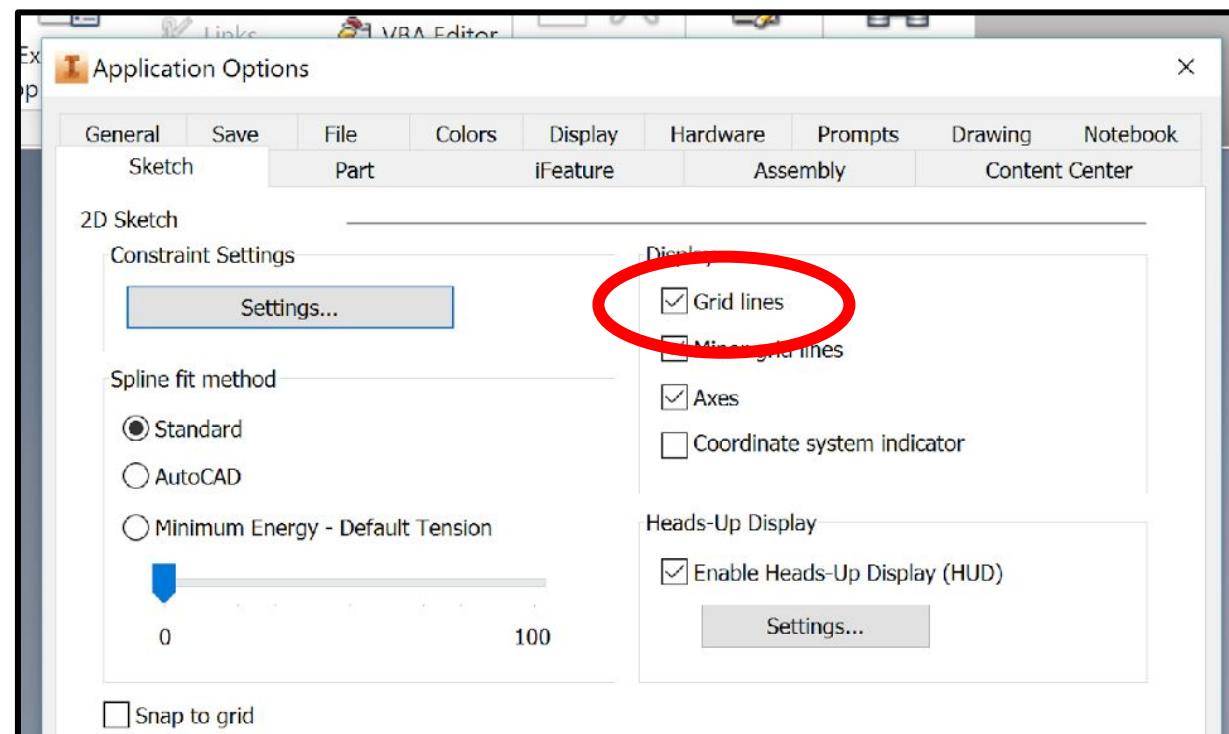
- When you are creating a sketch for the first time the origin planes will appear.
- Select one (doesn't matter which) to begin drawing on that surface.

WORKING IN THE 2D ENVIRONMENT

- While in the sketch environment you will be working in the 2D environment. So *do not try to rotate your sketch (i.e. move in the third dimension) because it will actually do it and tilt your work surface.*
- Before we start to draw there are a couple of things that are helpful to **prepare our workspace.** (*Not essential though for this purposes of this intro workshop let's just say that they are*).
- The first is **gridlines**, they give us a sense of scale for what we are drawing and visually useful if you're into that sort of thing.
- But more importantly are **projected geometries**. Projected geometries are elements in your sketch that you can use as reference entities, similar to work features but in the 2D environment. They can be used as anchor points for lines and other drawn elements.

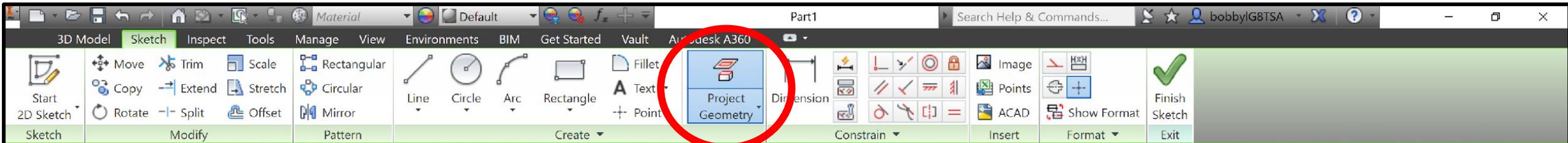
GRID LINES

- Tools > Application Options > Sketch tab > check the Grid Lines box

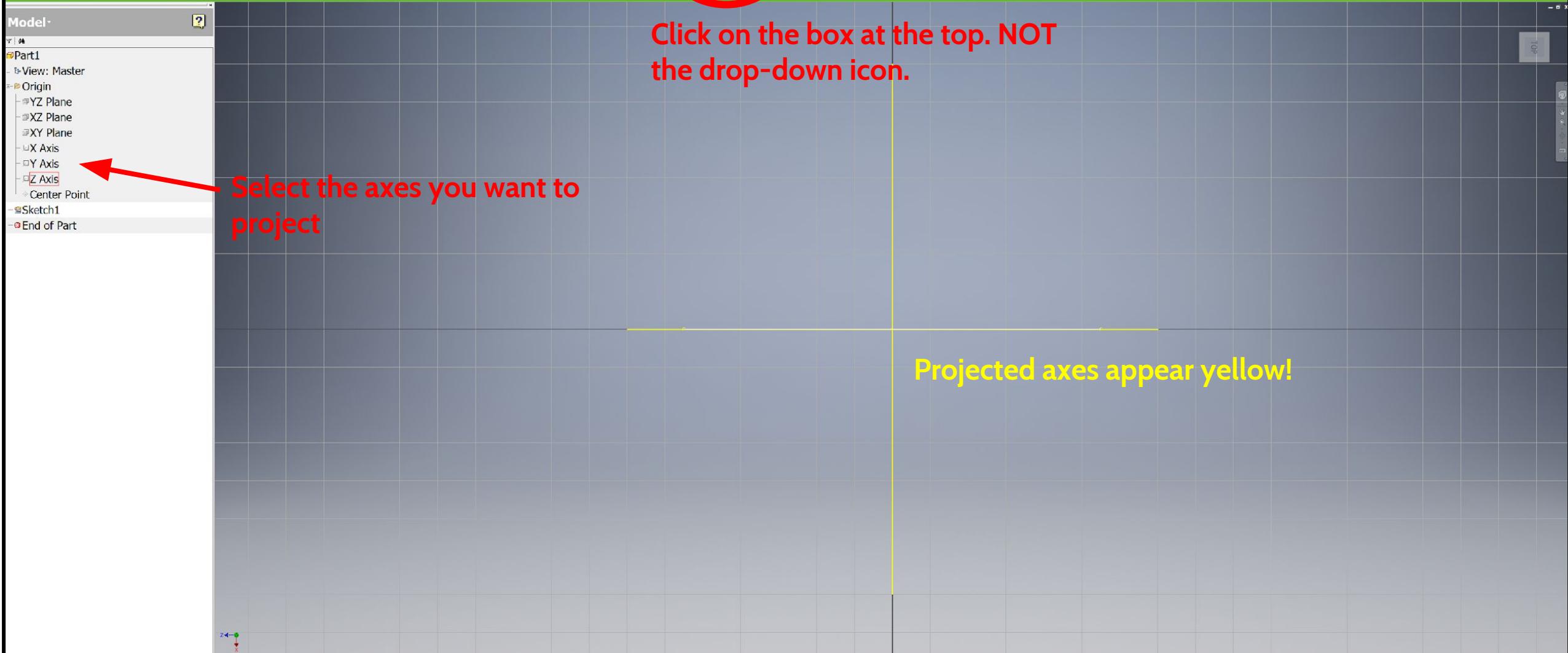


PROJECTED GEOMETRIES

- Projected geometries are elements in your sketch that you can use as reference entities, similar to work features but in the 2D environment.
- The element that can be projected are any lines or drawn elements, axes, or even entire cross-sections.
- To project a geometry, select the **project geometry** command.
- In this operation the command is asking you to select an element to project. You can select more than one which is why there is a small plus sign next to your cursor. *This type of multi-select operation appears in other commands as well so be aware.*
- When you care done, press **esc** or **right-click** and **select ok** to exit the operation.
- Any projected geometries will appear in **yellow**, like in the example of the **projected axes** which you should do to prepare your sketch environment.



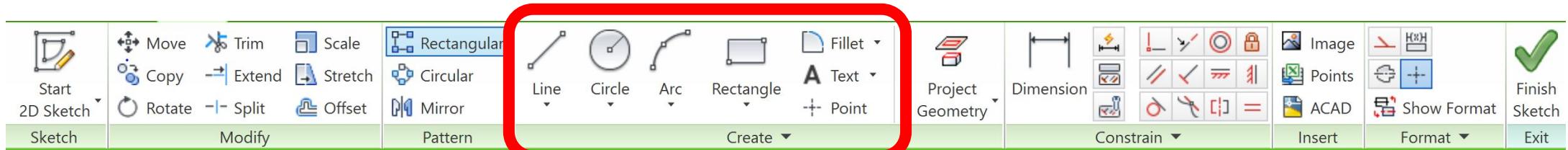
Click on the box at the top. NOT
the drop-down icon.



Projected axes appear yellow!



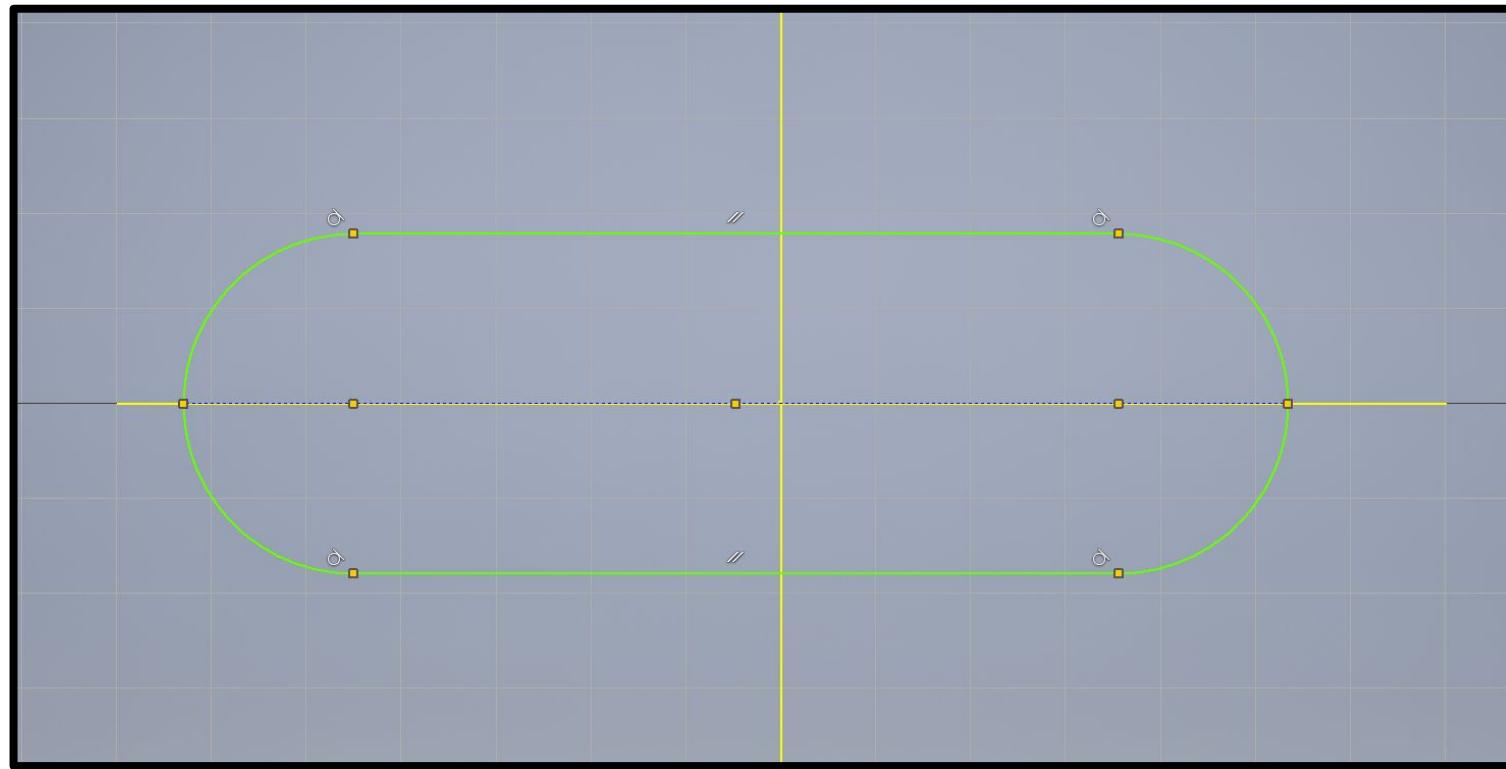
DRAWING TOOLS



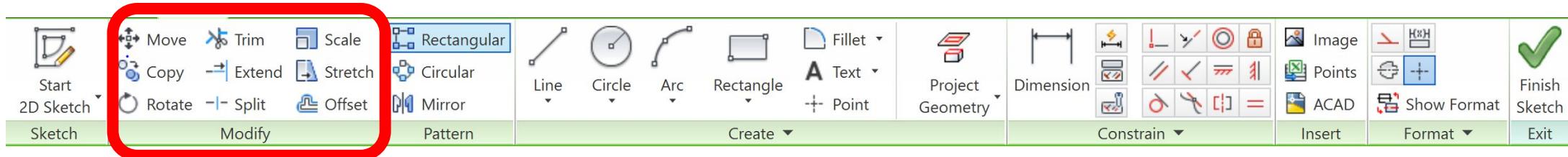
- There are basic drawing types: line, circle, arc, rectangle and fillet.
- There are many different ways to draw these elements, if you look in their drop-down menus you can find them, and combine them to make more complex shapes.
- The “ctrl+” command still apply so you can copy, paste and even undo.
- *If you're unsure of how to execute an operation hover over the option with your cursor to see a brief tutorial or refer to your command status (bottom-left).*

PRACTICE SKETCHING

- You can be creative when sketching, let's try to create an oval/skateboard shape using three different methods:
- 1. Slot Polygon (underneath the rectangle drawing tool).
- 2. Use only some combination of line, circles and arcs.
- 3. Use a rectangle and use the fillet modification to round out the corners.



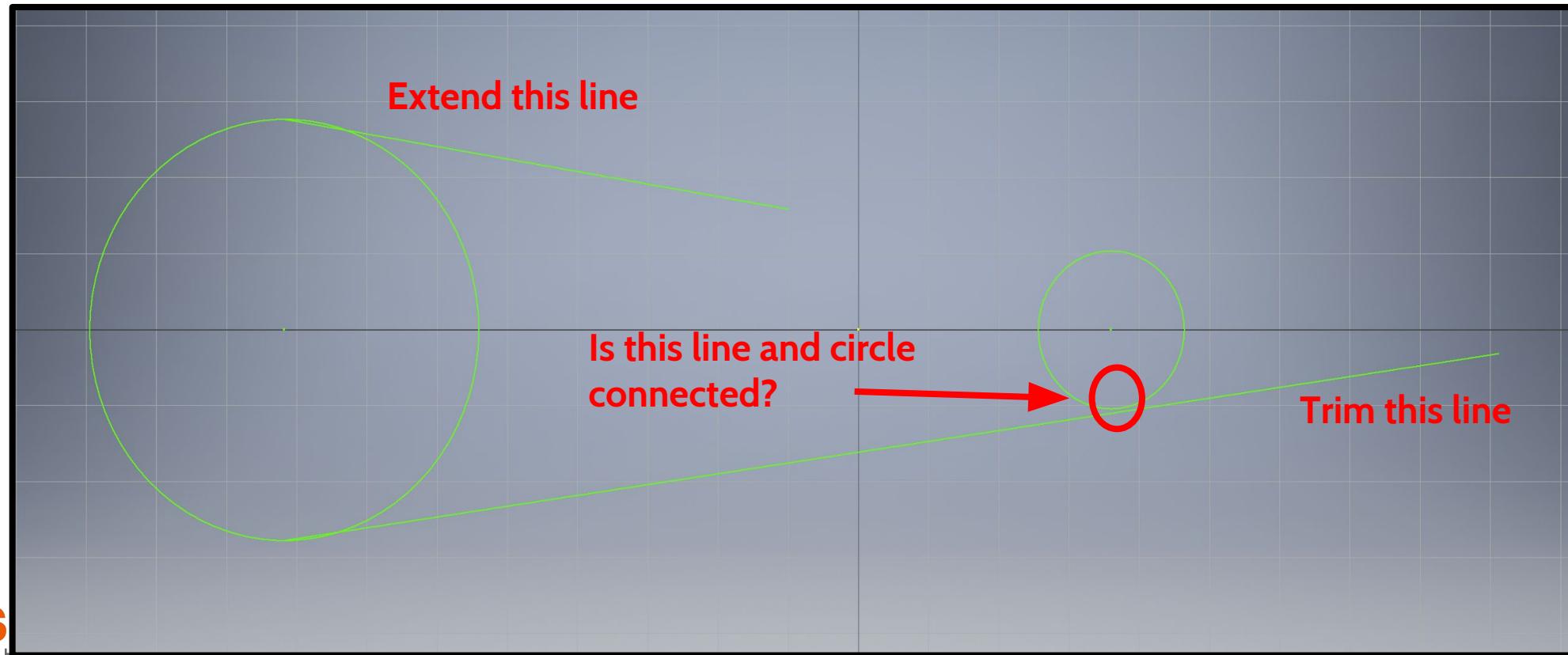
MODIFICATION TOOLS



- Sketched can also be modified if you cannot get the desired **shape** using just basic drawing elements.
- In the next workshop example we will open the **trim_and_extend** file to demonstrate two of these modification tools: the trim modification and extend modification.
- Another important lesson that can be learnt from the example is never to draw or drag elements randomly because this can lead to discontinuities in your sketch that you might be able to pick up on with just your bare eye.
- *The file can be found in the “O2_Part” folder in the downloaded materials.*

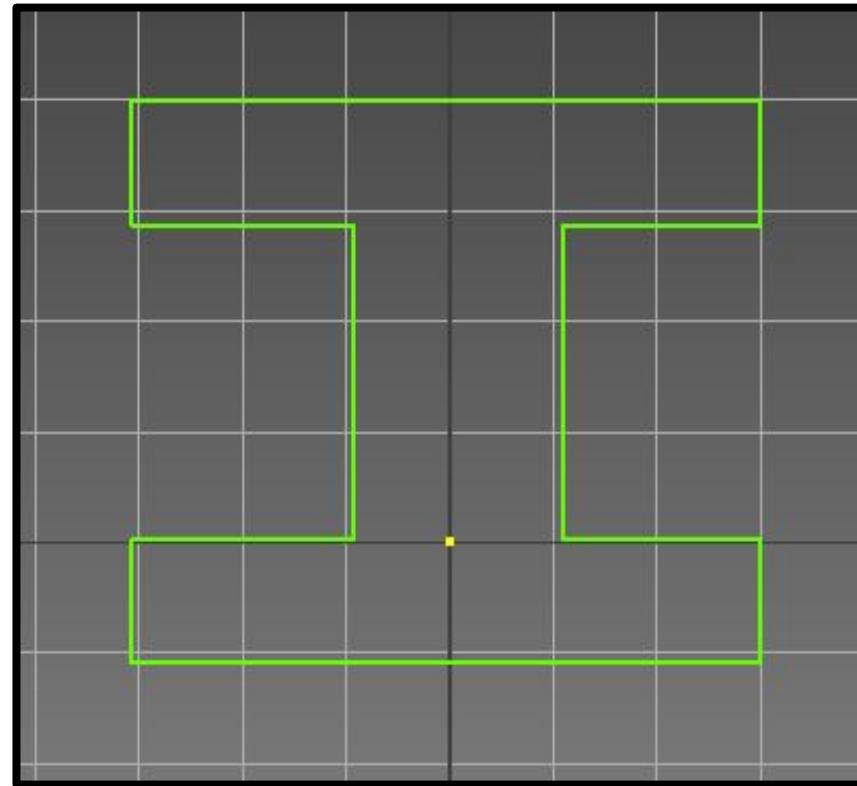
TRIM AND EXTEND EXAMPLE

- You are editing this sketch. *Do not start a new one because that will just layer another sketch on top of it.*



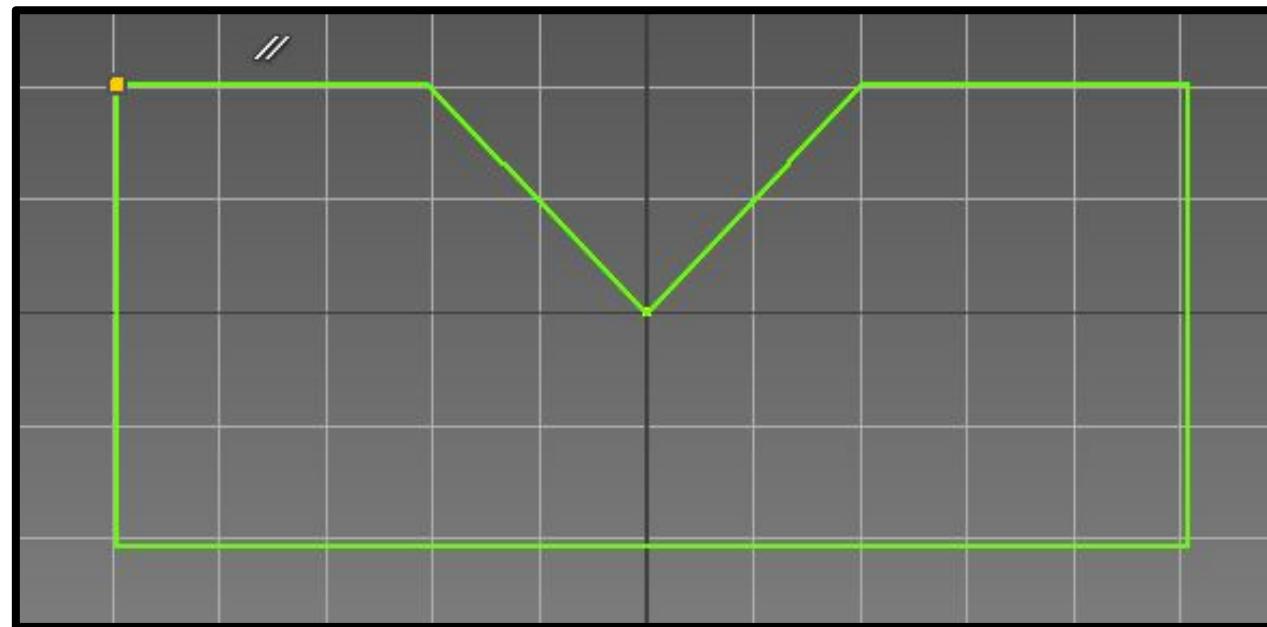
I-BEAM EXAMPLE

- Try to draw the I-beam as shown. You just need to get a decent **shape** for now.
- Are you trying to centre your sketches to the origin? Do you think it is a good idea to do so?



NOTCH EXAMPLE

- Draw the notch as shown. You just need to get a decent **shape** for now.
- You now know about the drawing tools and modification tools, how do you want to approach this sketch?



CONCLUSION

- We looked at working in the 2D environment to produce profiles (our sketches).
- We prepared our workspace before we started sketching by applying grid lines and projecting our geometries. Remember that grid lines can be forever but projected geometries are not so you need to do it again each time.
- We learnt how to use drawing tools and modification tools to sketch the **shape** of a profile. There is no wrong way to sketch, how you want to approach the design is the correct way.
- Something to think about:
 - When you selected a plane to draw on you probably chose one at random, *this is fine*. But if we start to work on a project that included multiple parts, would you define an appropriate orientation for each part?



DIMENSIONS & CONSTRAINTS

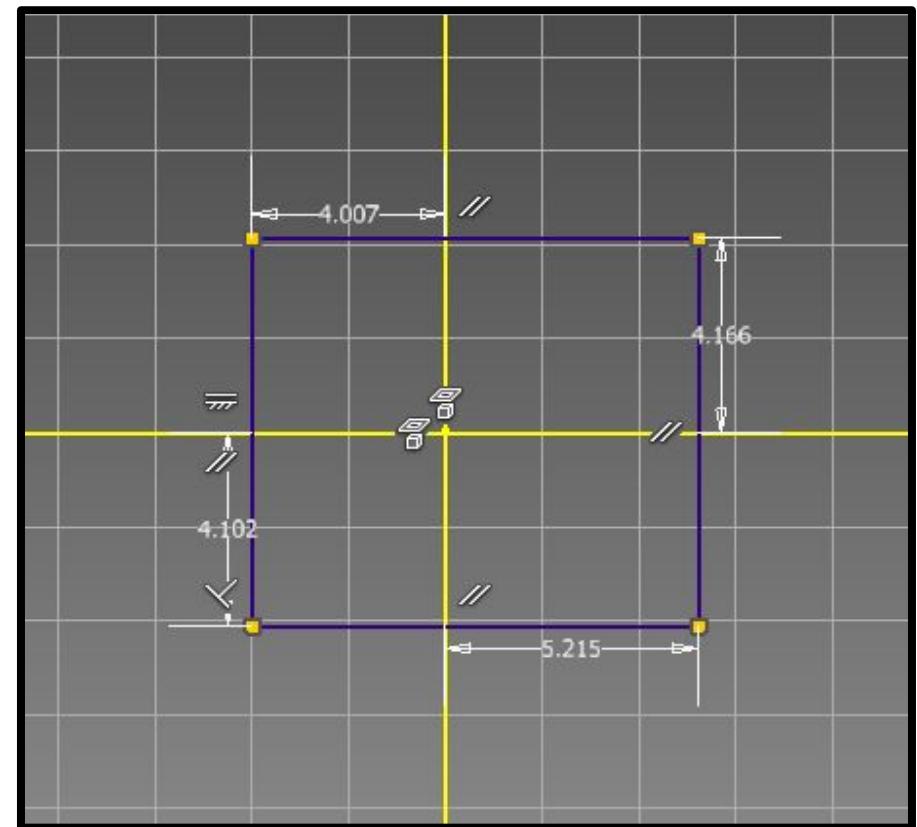
LESSON 3

MOTIVATION

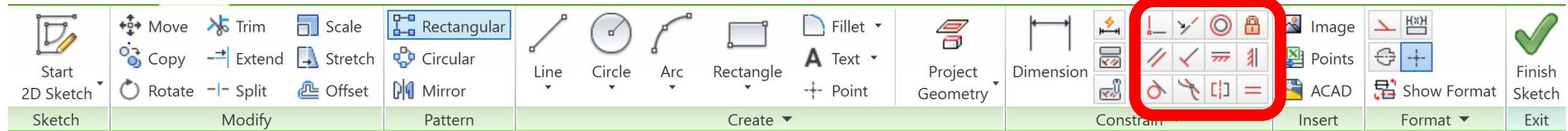
- So far we have the skills to taught the skills to draw general shapes. But that probably isn't going to be good enough when we are designing components that need to be an exact shape or size.
- You might also have noticed that the lines in our sketched so far appear green, and they can be freely moved around. This could be problematic when we're designing something and accidentally shift something.
- So now our aim is to be able to produce a sketch that is fixed, has one unique or exact solution and can be replicated by anyone that sees it.
- We can do this by introducing dimensions and constraints; we will be looking at constraints first.

WHAT ARE CONSTRAINTS?

- Constraints are relationships between two entries, and these can be applied to the elements in a sketch.
- As you sketch, some constraints may have already been automatically applied. *This is the Inventor program trying to think one step ahead.*
- For example, if you draw a horizontal or vertical line a symbol will appear indicating that the associated (horizontal or vertical) constraint has been applied.
- A sketch that has been **fully constrained** will appear **dark blue**. As in the image here.



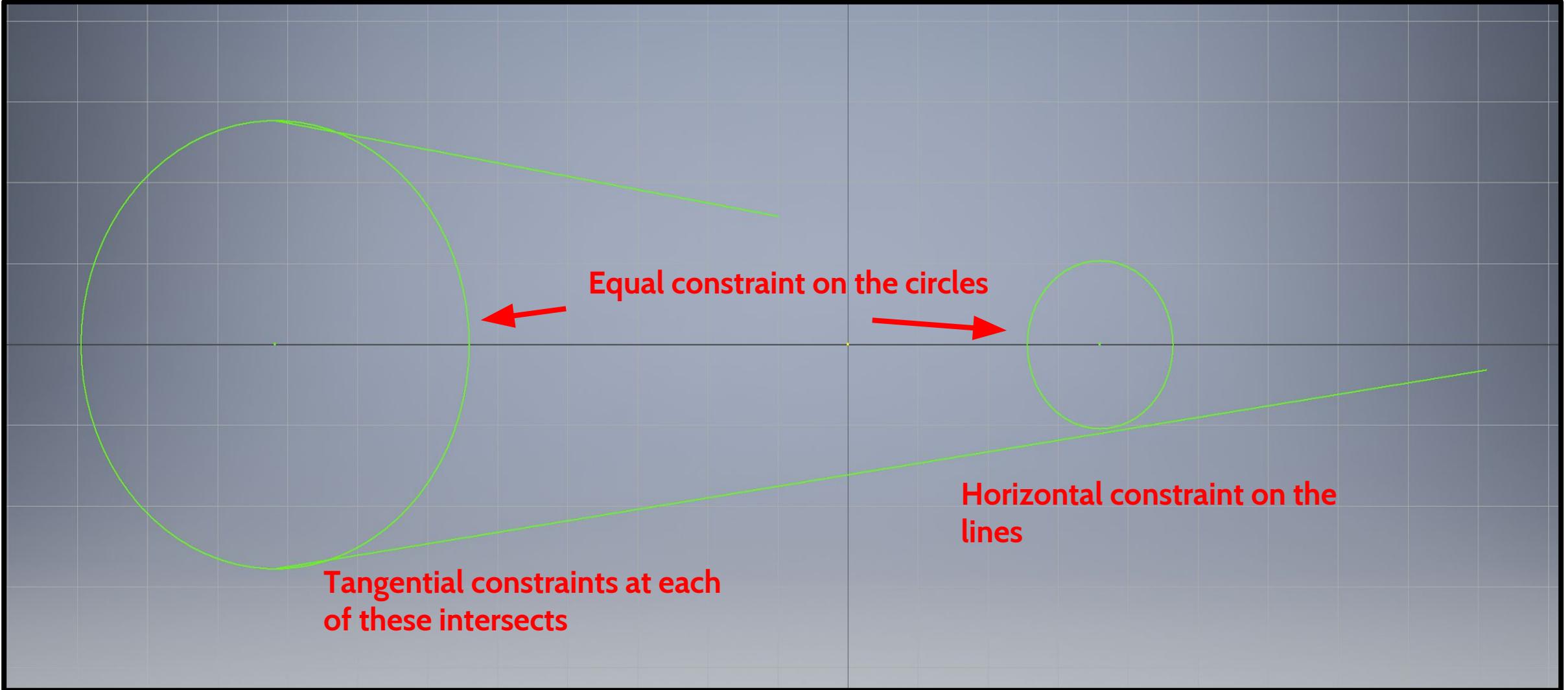
TYPES OF CONSTRAINTS



- There are a number of different constraint **types**, so each are applied in a different way (you will need to select different elements for each) and some even have unique characteristics.
- For example, the **symmetry constraint** forces two elements to mirror each other reflected about an axis. To apply this, you will need to select the first element then the second element you want to reflect, and then the mirror axis.
- Some interesting characteristics:
 - You cannot select two elements on the same side of the axis (or crosses the axis) or else the program might get confused.
 - And after your first symmetry constraint is applied the command will remember the mirror axis you first selected so any subsequent constraints will follow that until you exit the command.

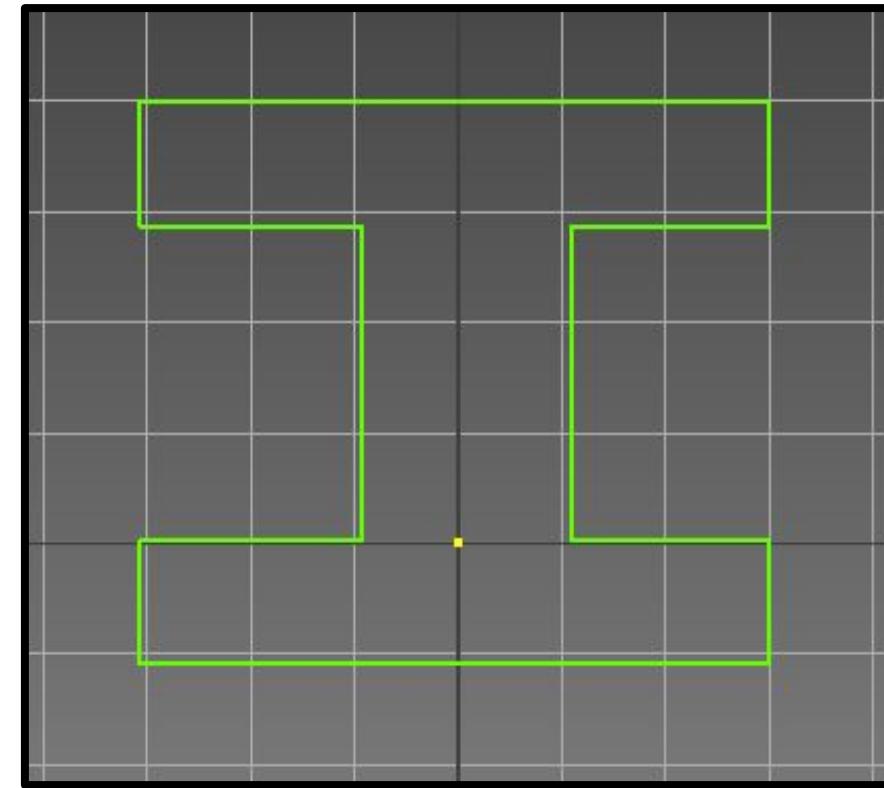
TRIM AND EXTEND EXAMPLE REVISITED

- Let's go back to our `trim_and_extend` file and see if we can apply to see how many constraints we can apply to it.
- Does the sketch begin to resemble any of the sketches we have done previously?
- The important lesson here is that it doesn't really matter if you **sketch badly**, if anything it's advised, because when you apply your constraints it won't matter.



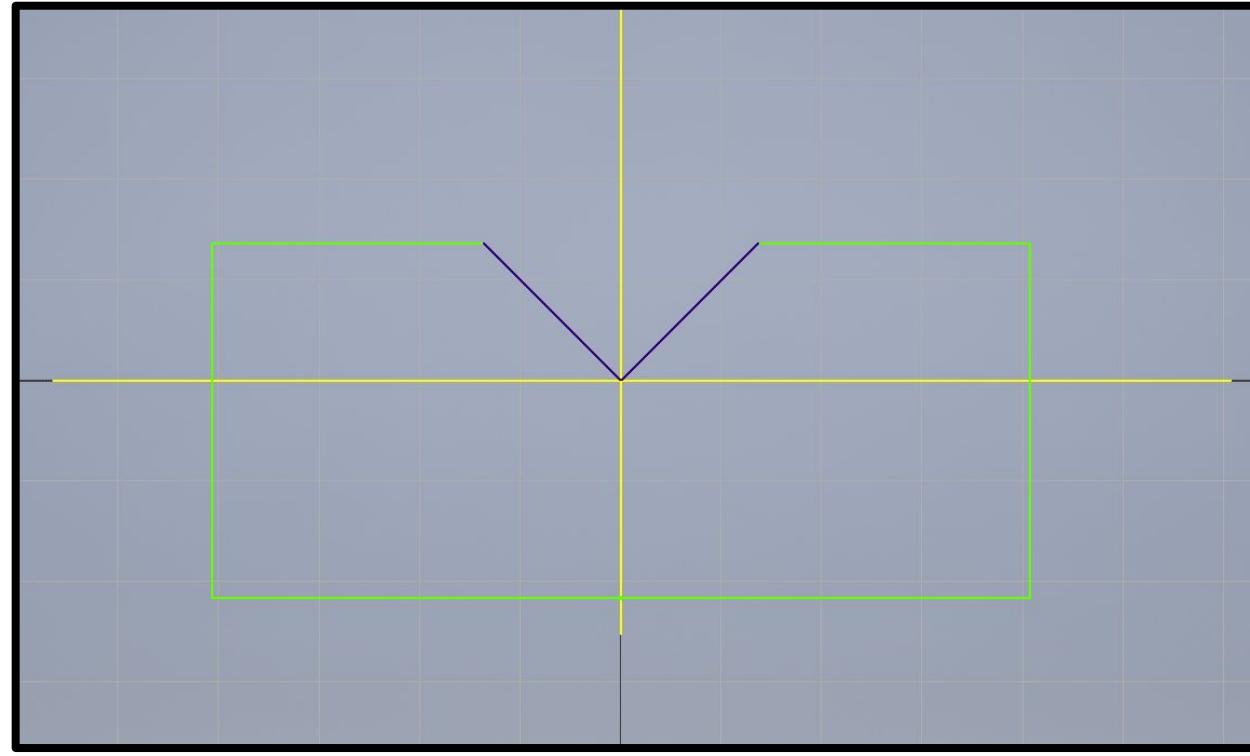
I-BEAM EXAMPLE REVISITED

- Let's go back to the I-beam exercise and apply our constraints.
- Try to think critically to determine where the constraints need to be applied:
 - You can show all applied constraints using **F8**, and hide all constraints using **F9**.
 - You can display the degrees of freedom by right-clicking on the sketch and selecting the option. Degrees of freedom will highlight where and how the sketch can translate, apply a constraints to restrict its movement.

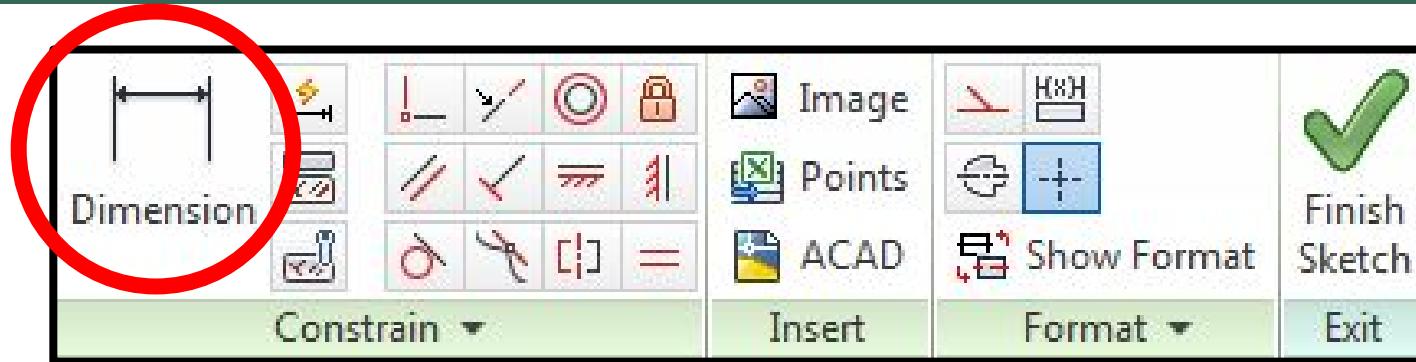


NOTCH EXAMPLE REVISITED

- Let's go back to the notch exercise and apply our constraints.
- Here's a question:
 - Can you get your sketch fully constrained by applying only constraints?
- The **blue lines** in the image indicate that only some lines have been **fully constrained** using only constraints. *That's my attempt can you do better than me?*



DIMENSIONS



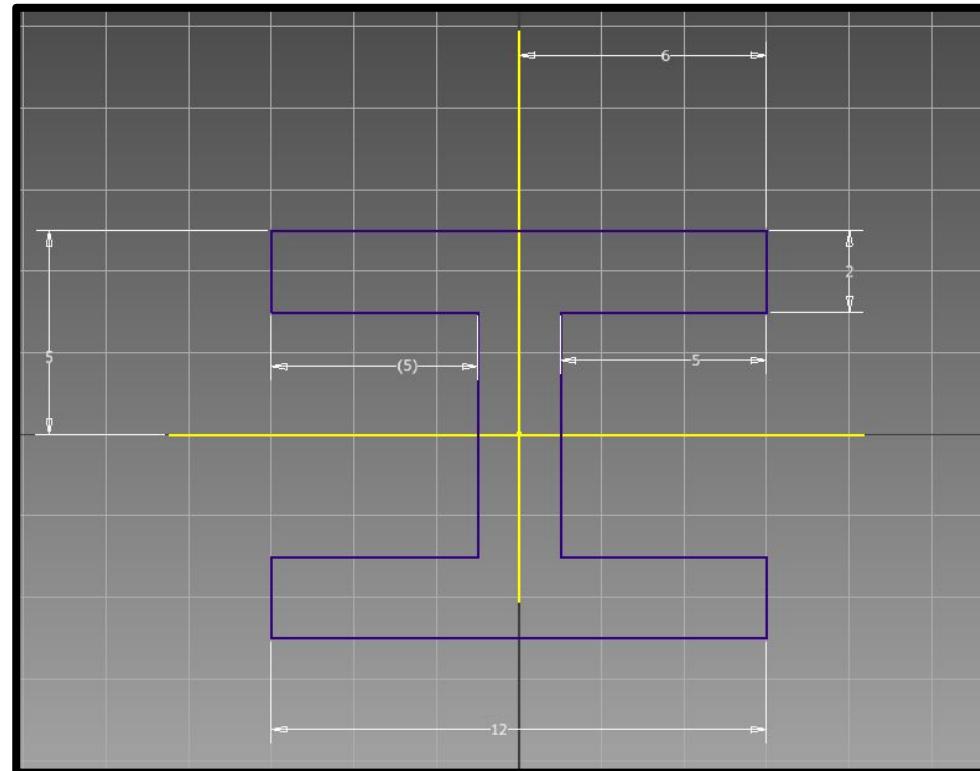
- It is almost impossible to fully constrain a sketch with just constraints, you need to also apply dimensions.
- Dimensions are the numerical value for how long, wide, round, etc. your profile needs to be.
- In the **fork_example** we edited a dimension on a sketch, and this changed the underlying diameter value for that 3d component.
- Dimensions are also a constraint technique because it ultimately restricts how our model appears, but more importantly dimensions provide people with information about your design.
- Try to imagine manufacturing a part without being given any dimensions?

DIMENSIONING TECHNIQUES

- Any horizontal or vertical dimension can be applied selecting the line.
 - Do not hold down your mouse button whilst in the dimension command.
 - The dimension value will follow you mouse, so you just need to left-click to place it.
- A diagonal dimension can be applied by selecting the line twice.
 - *Leave a slight pause between clicks, do not double-click.*
- A distance dimension can be applied by selecting two lines or points.
- A curvature dimension can be applied by selecting a curved line
 - You have to option of using a diameter or radius dimension.
 - Right-click to open the options from the menu.

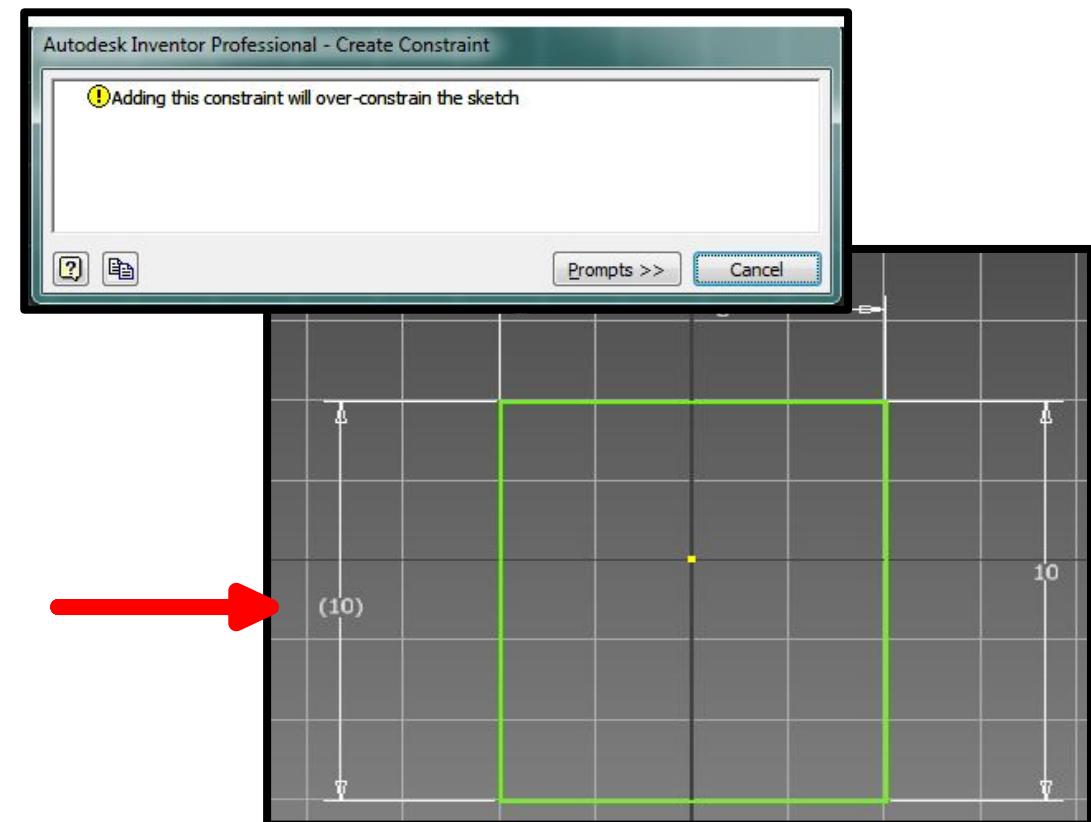
PRACTICE DIMENSIONING

- Apply some dimensions to your I-beam and notch example, to get them fully constrained.
- Notice on the bottom-right there is a **number for dimensions needed**.



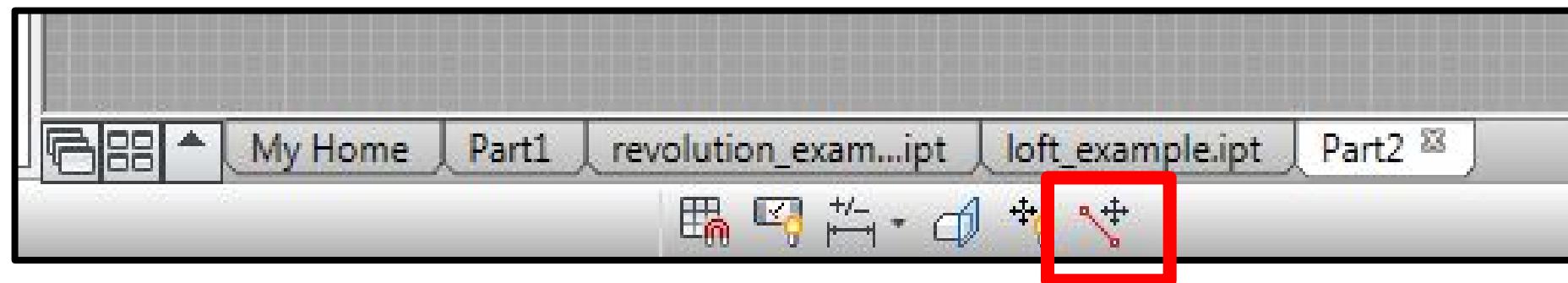
OVER-CONSTRAINED AND DRIVEN DIMENSIONS

- You cannot over-constrain a sketch, over constraining will cause a dialog box to appear.
- Driven dimensions do not constrain the sketch, rather they are dimensions that cannot be altered because of the constraints currently in place. They are enclosed in **bracket parentheses** to distinguish them from normal parametric dimensions.



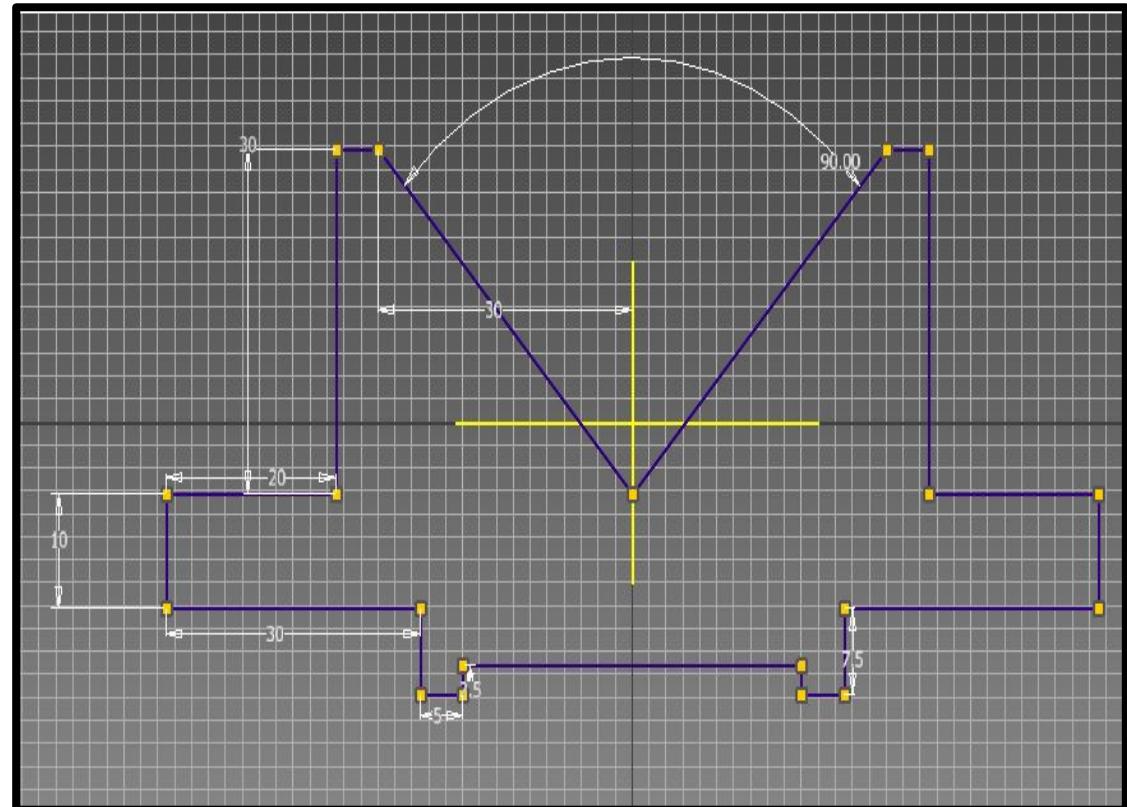
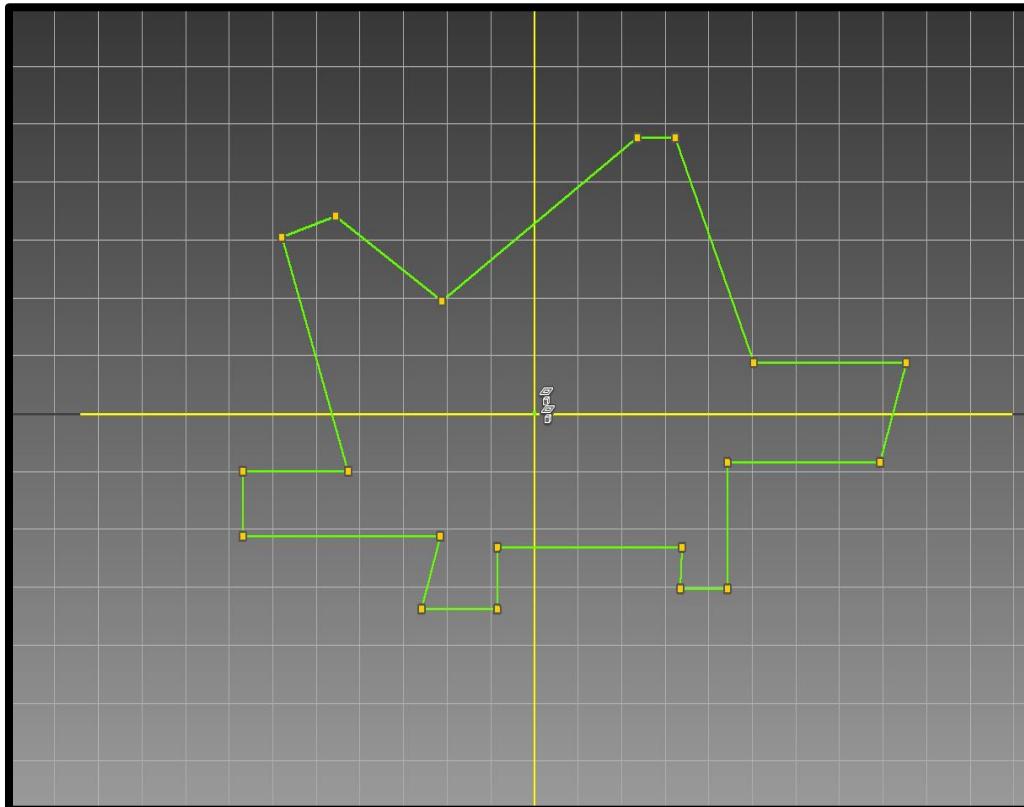
RELAXED CONSTRAINTS

- Constraints can be relaxed, if you select the relax constraints icon or hit F11
- This will temporarily break any constraints for you to reconfigure the sketch freely



WORKSHOP CHALLENGE 1

- Open **sample_file_1.upt**, found in “O2_Part” folder of the downloaded materials.
- HINT: Use project geometry and symmetry constraint

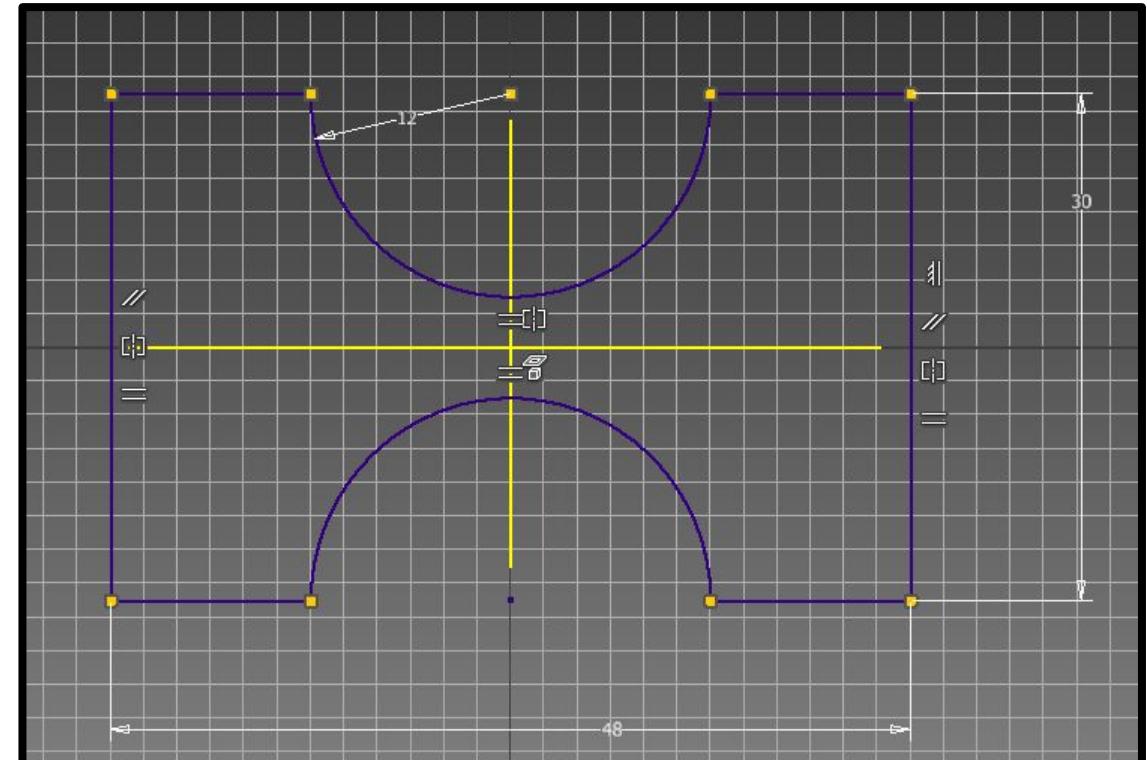
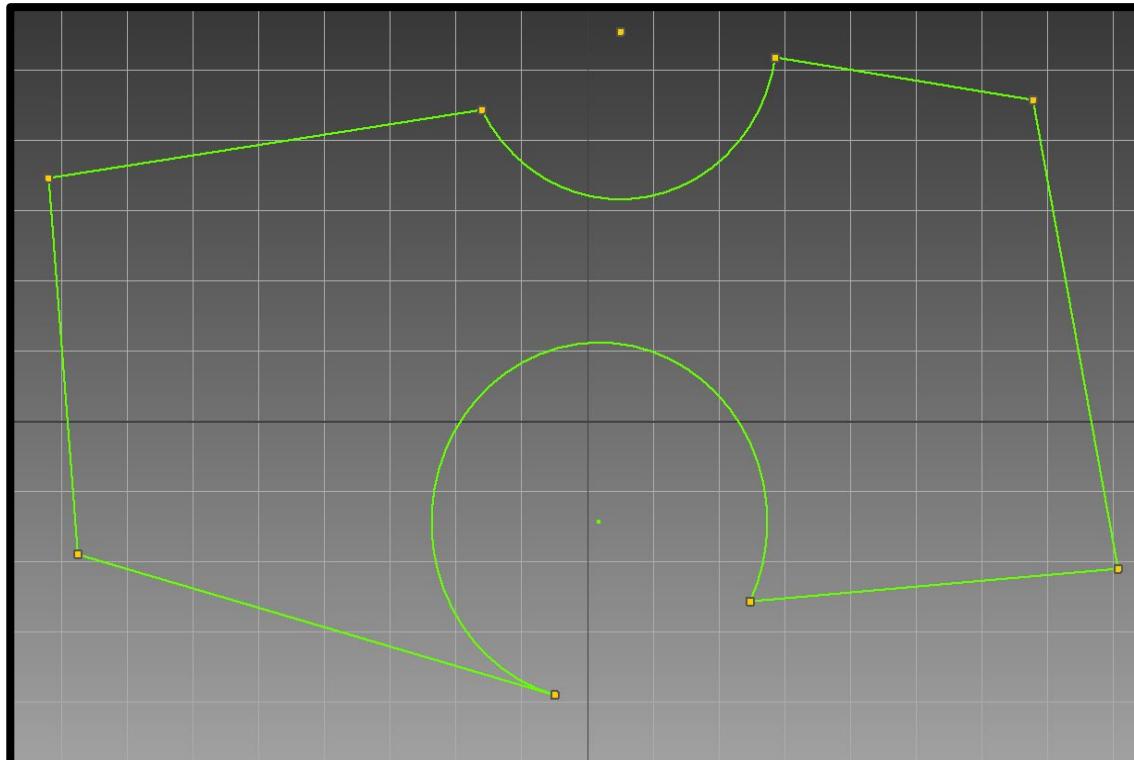


WORKSHOP CHALLENGE 1 REVIEW

- We will do a demonstration of how to constrain and dimension this profile.
- A few tips that can be helpful:
 - Notice that the sketch is **symmetric**, so you can constrain one side of the sketch and then just apply a symmetry constraint about the axis to halve your workload.
 - When constraining be willing to move around lines to maintain the general shape of the sketch.
 - When dimensioning, the sketch can be prone to doing strange things. Simply undo the operation and apply your dimensions else where to that the general shape is at least maintaining.
 - Remember to also contain your sketch to the workspace (to the origin) so that it is completely fixed.
- *There is a strange dot in the centre of the sketch, delete it.*

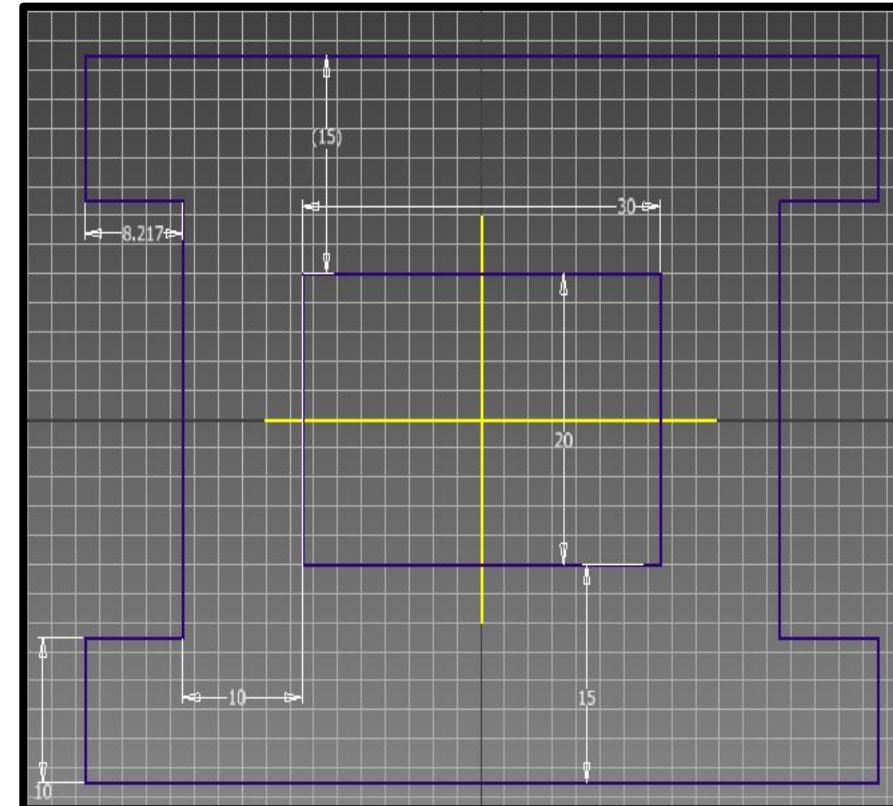
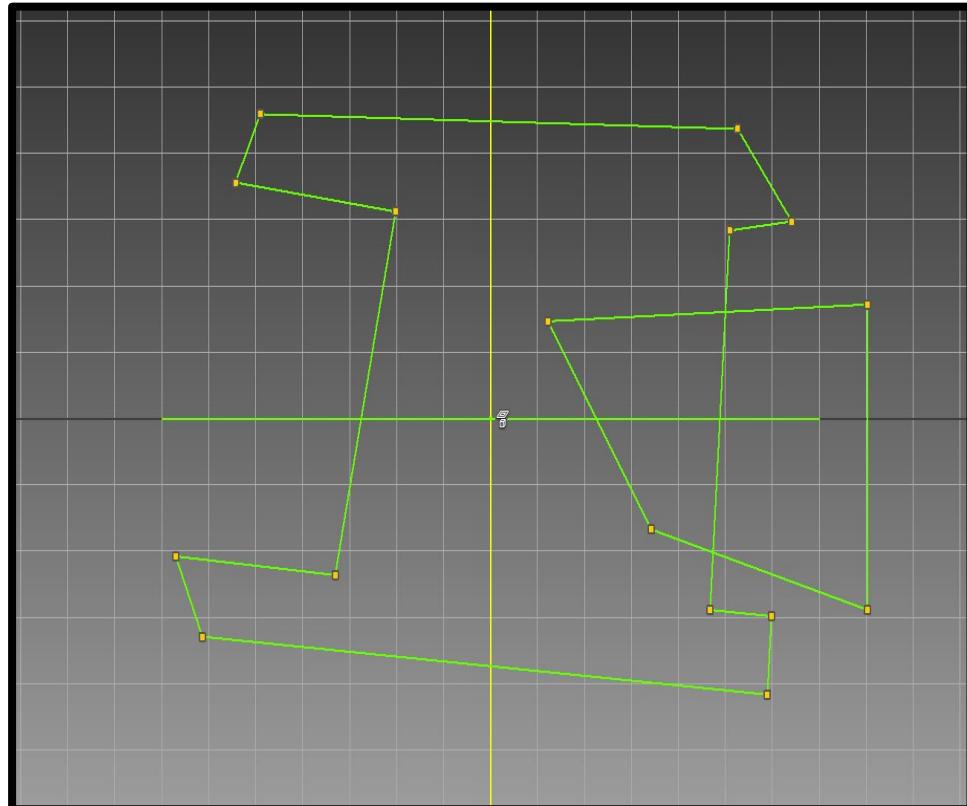
WORKSHOP CHALLENGE 2 (OPTIONAL)

- If you finish early, or want some extra practice.
- Open **sample_file_2.ipt**, found in “O2_Part” folder of the downloaded materials.



WORKSHOP CHALLENGE 3 (OPTIONAL)

- If you finish early, or want some extra practice.
- Open **sample_file_3.ipt**, found in “O2_Part” folder of the downloaded materials.



CONCLUSION

- We looked at dimensions and constraints. These are relationships that we can put onto our sketches to produce a **singular solution** that is fixed in space.
- Our sketches can now be replicated by anyone, we did this when we did the workshop example.
- You now have an understanding of the different types of constraints and how to apply them.
- You now have an understanding of dimensions and the different techniques used to apply them.



LUNCH TIME
MEET BACK IN ONE HOUR.



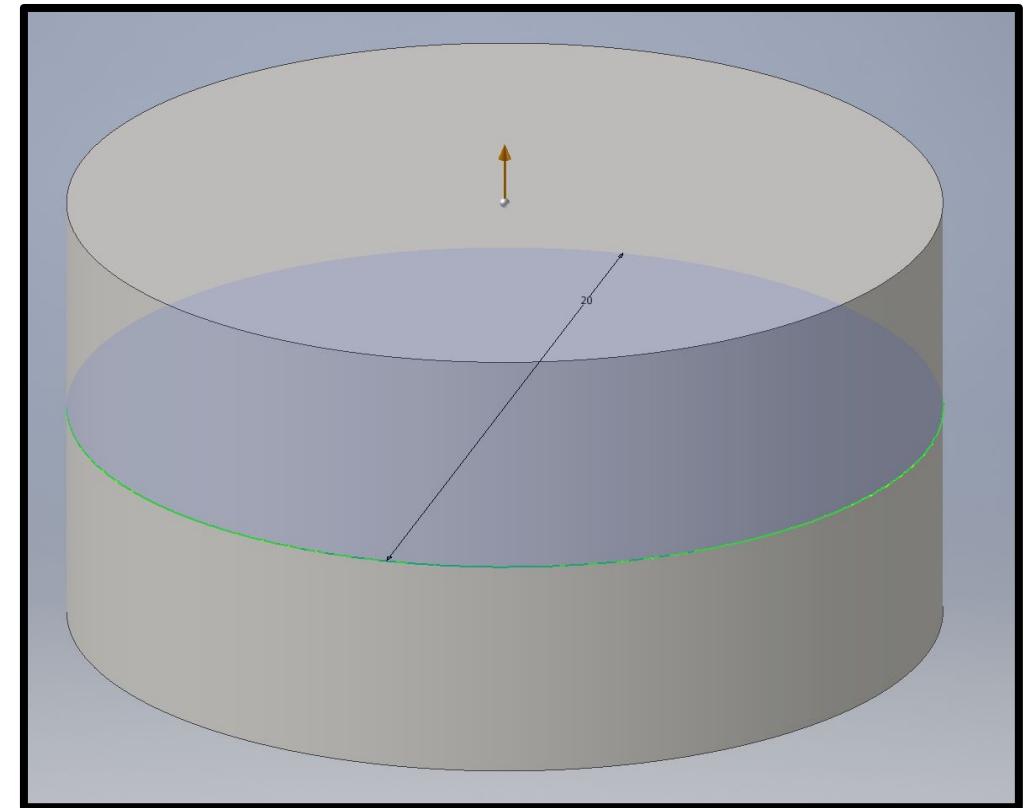
BASIC 3D MODELLING

LESSON 4



MOTIVATION

- Now that we have established the fundamental skills, this lesson will introduce 3D modelling concepts.
- We will look at base (primary) and secondary 3D features, as well as 3D modification tools.
- Concepts which you may find similar to modelling in the 2D environment during sketching.

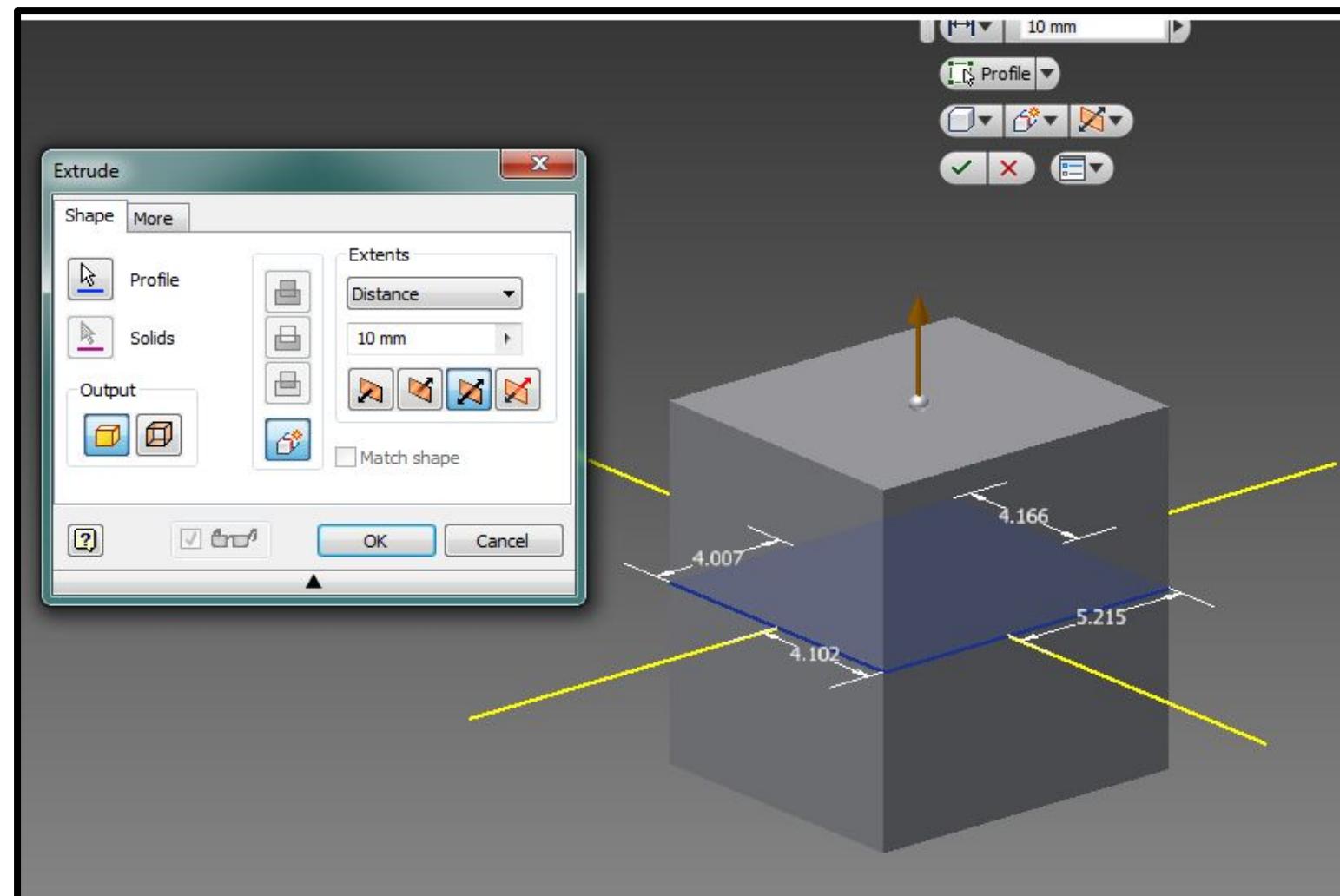


SOLID BASE FEATURES

- The solid base feature is the first 3D element you create.
- All subsequent features you make could be based off of this main solid, but this might not necessarily be the case.
- The types of 3D features you can create are: **extrusion, revolve, loft, sweep, coil**.
- Each require different elements to execute.

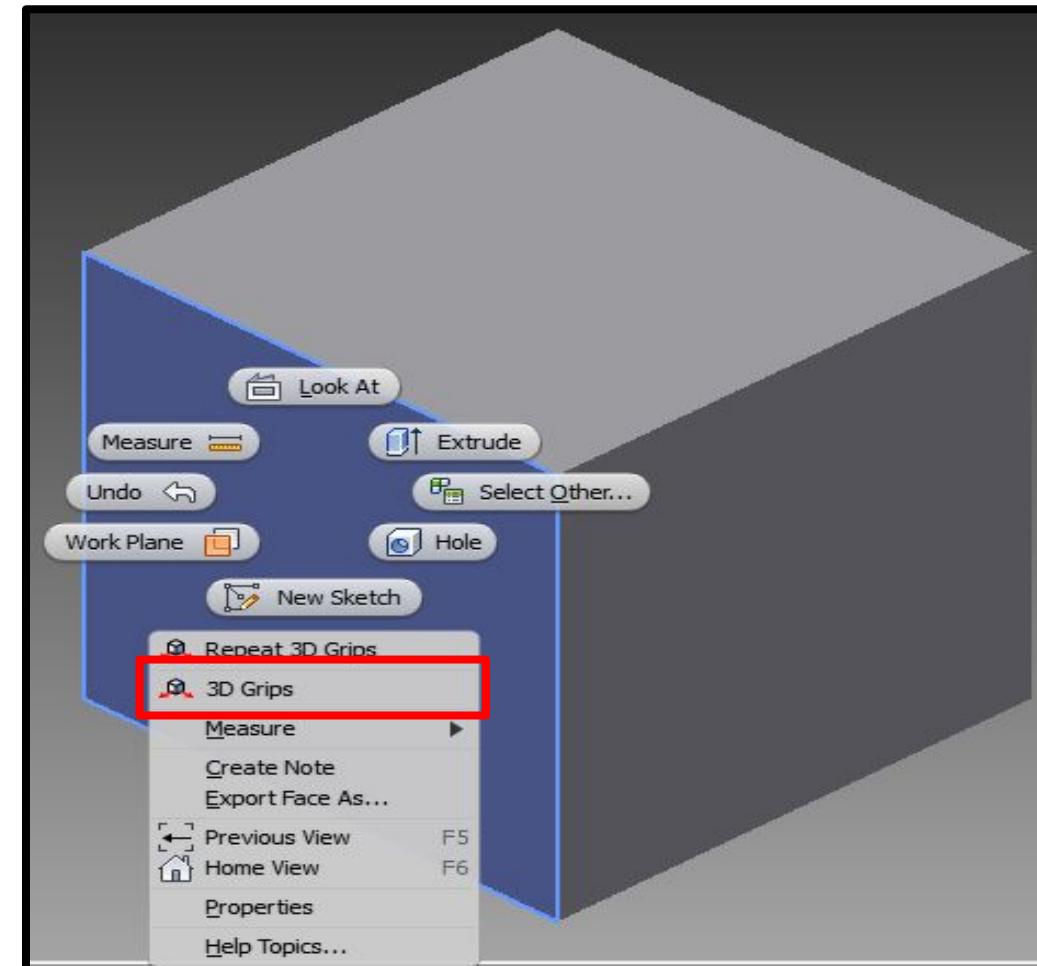
EXTRUSION

- Extrusion is the most basic and most **common** type of 3D component, because it is **simple** to use.
- Let's do a demonstration creating a cube solid.
- *Extrusions require:*
 - *One 2D profile*
 - Take note of the **extent** options, and **taper** options under the 'more' tab.



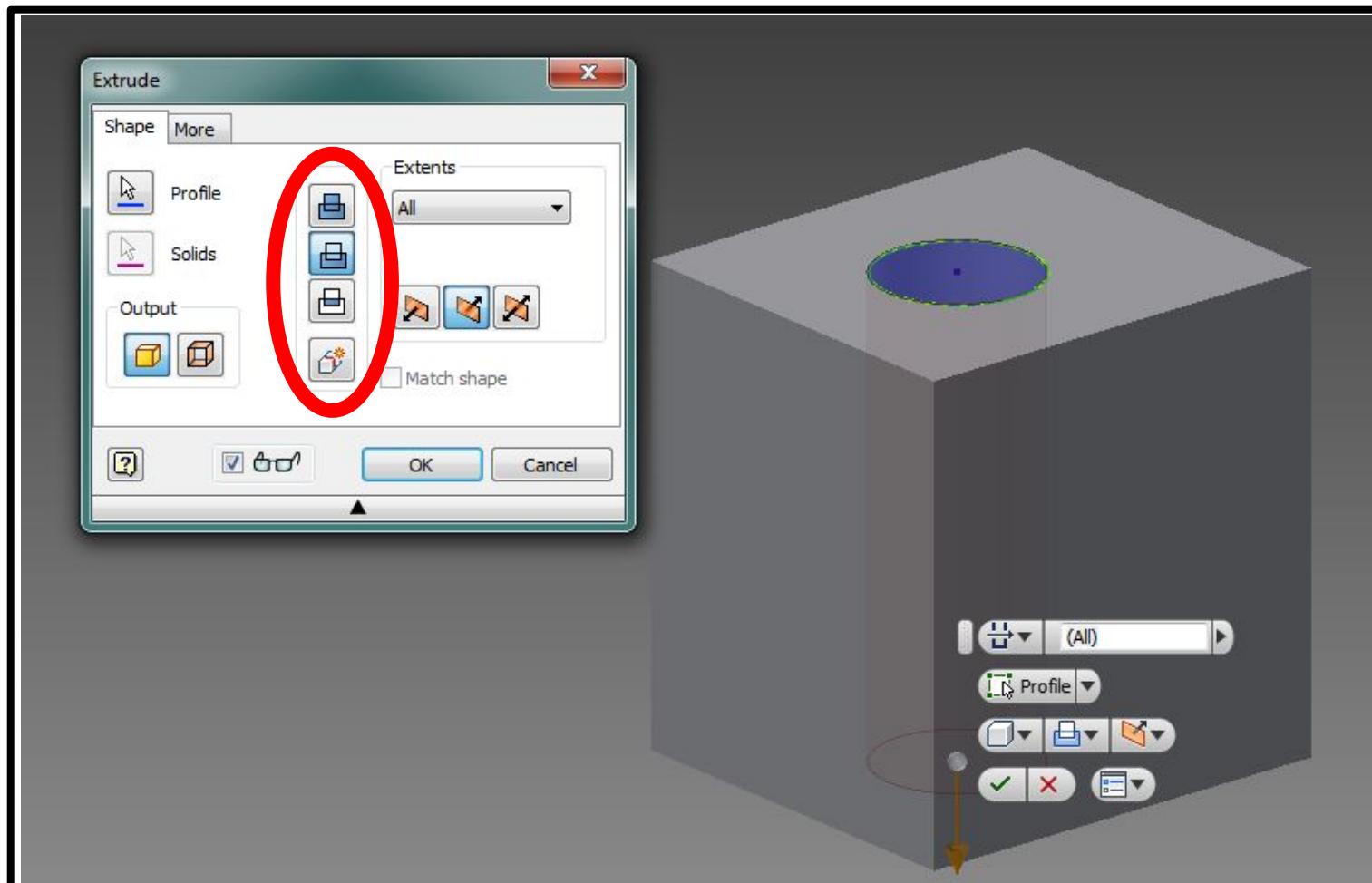
3D GRIPS

- On a side note, there is a 3D grips option which is very similar to relax constraints in the 2D environment.
- 3D grips enables you to **partially freeform** your model by manipulating the parametric values (dimensions) of your model. *Shape stays the same.*
- Right-click on your model to find this option in the menu.



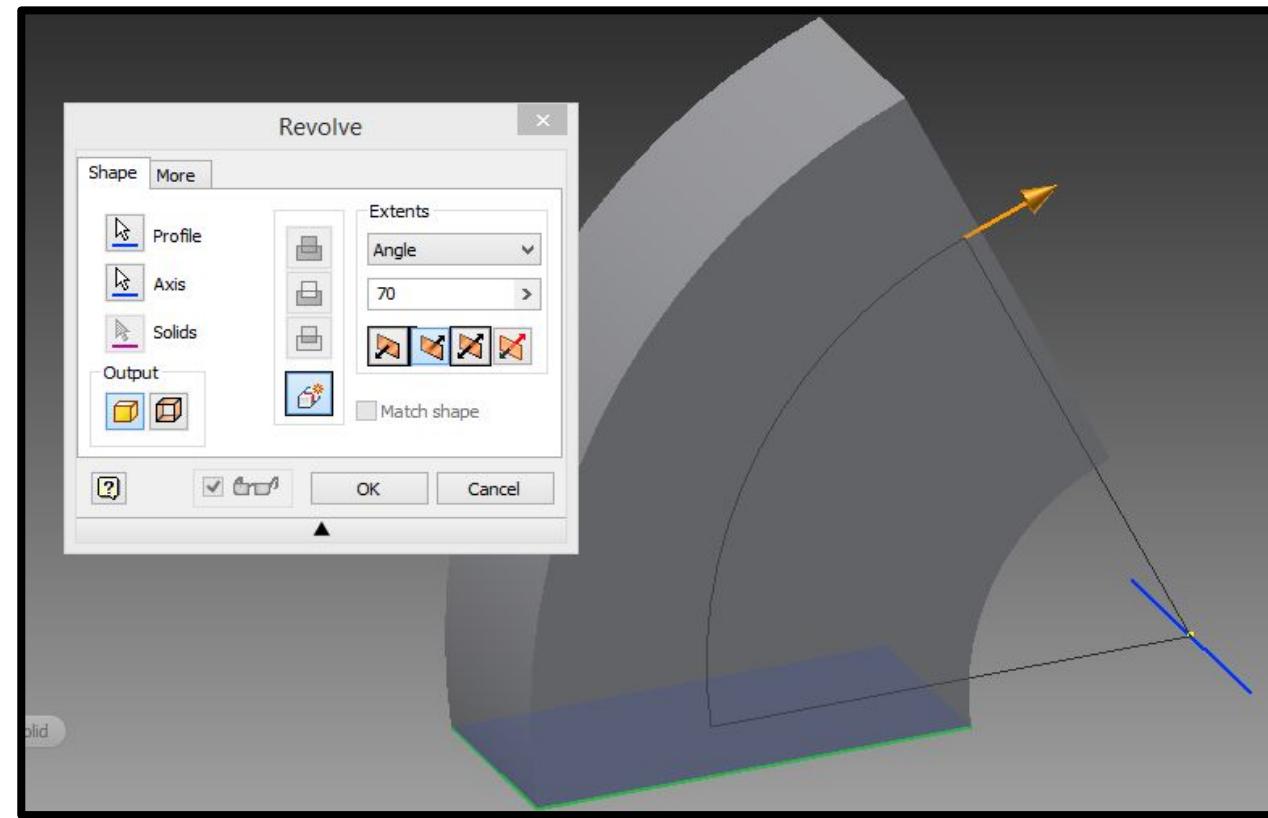
SECONDARY EXTRUSION FEATURE

- Secondary extrusions build off your base feature.
- To demonstrate this, let's create a second extrusion (hollow section) on our base extrusion.
- Take note of the type of solid feature we are creating: **join, cut, intersect, or new solid**.



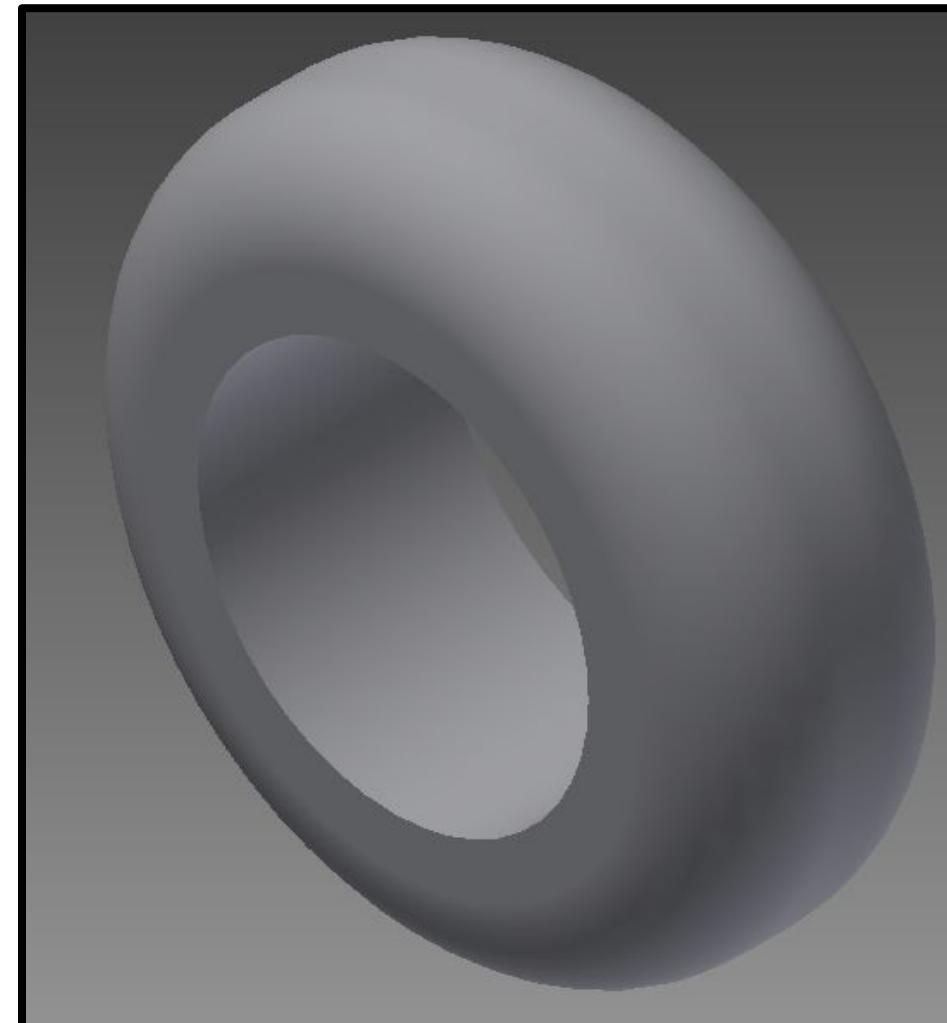
REVOLVE

- The revolve method takes a cross-sectional profile and revolves it about a reference axis.
- *Revolve requires:*
 - One 2D profile
 - One reference axis
- Let's demonstrate this by creating a donut shaped solid.
- *Can you use the revolve method to create a sphere? What would the cross-sectional profile look like?*

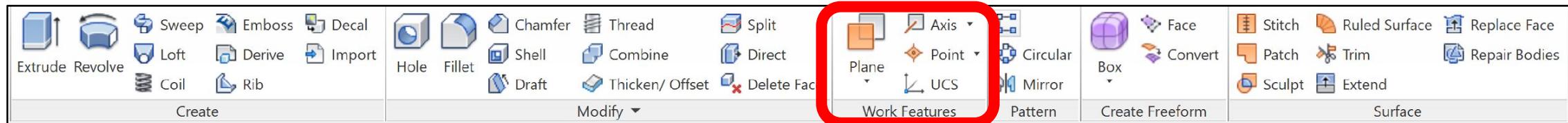


TWO WAYS TO CREATE CHALLENGE

- We can model 3D objects in different ways.
- Try to create this wheel using:
 - 1. Revolve tool
 - 2. Extrusion and the fillet tool
- The two models do not have to be identical so don't worry too much about having the exact dimensions.
- *Fillet is a type of 3D modification tool, we will speak more about this later.*



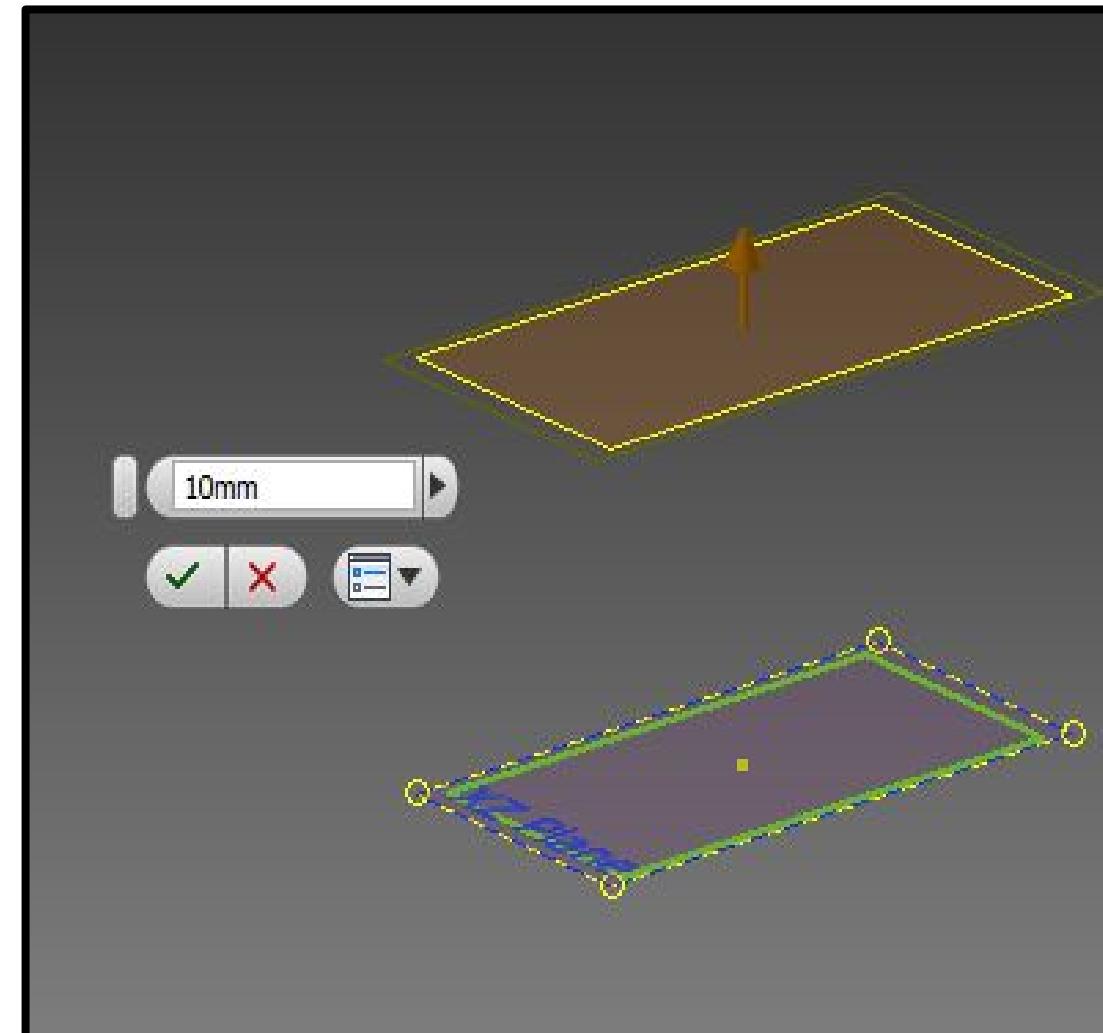
CUSTOM WORK FEATURES



- We will take a quick aside, before we press on with 3D modelling.
- Until now we have only seen the **origin work features**, and how we are able to use them as reference elements to create our sketches and 3D objects.
- But what if we needed to create something that was not supported by these work elements because it was away from the origin?
- Our solution to this problem is using **custom work features**, that is work features that you designate yourself.

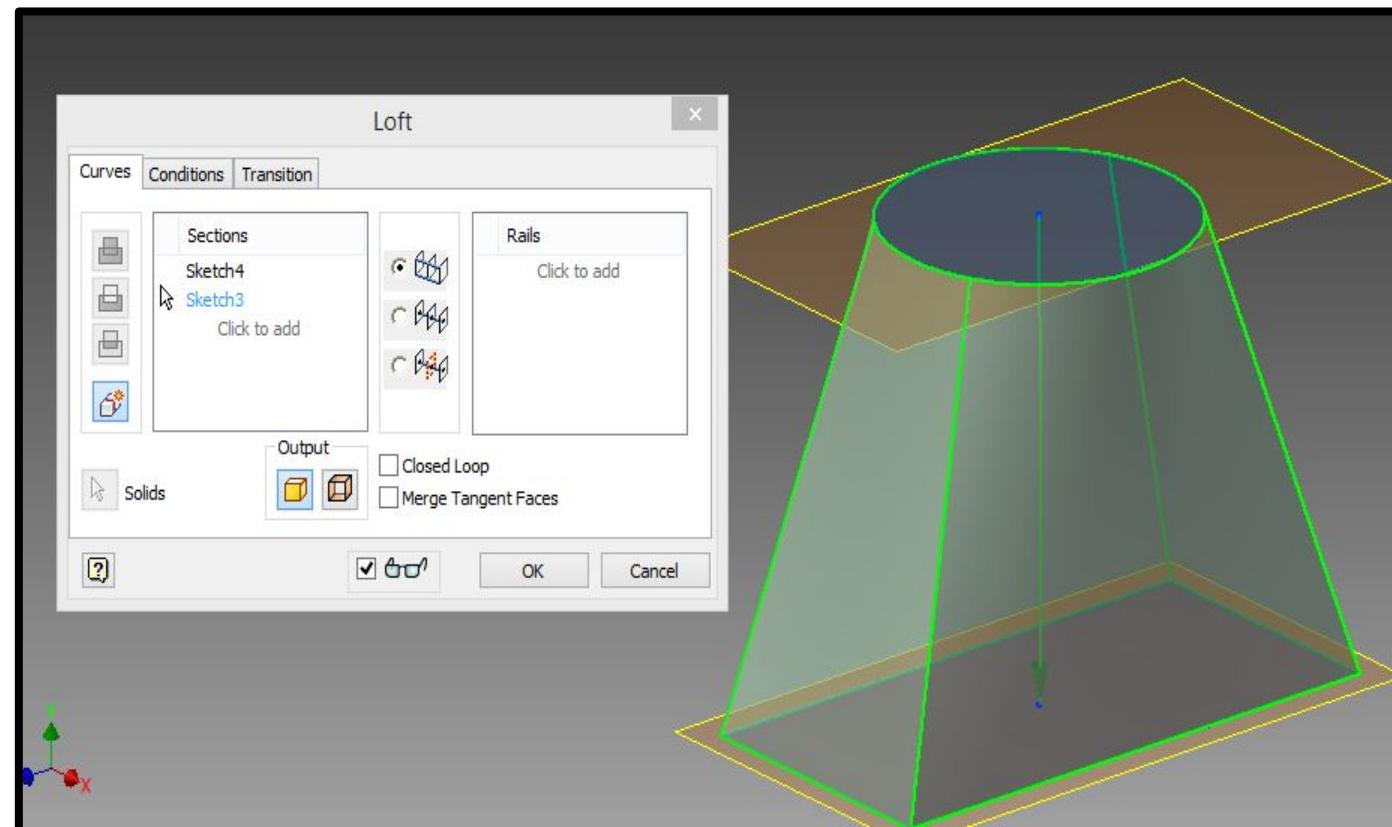
CUSTOM WORK PLANES

- For the next couple of examples we will need to create custom work planes.
- We will use the **offset work plane** and the **mid-plane** techniques to achieve this.
- The placement of offset work planes can be done freely or parametrically.
- The **mid-plane** technique will place a plane ***equidistant*** between both reference planes.



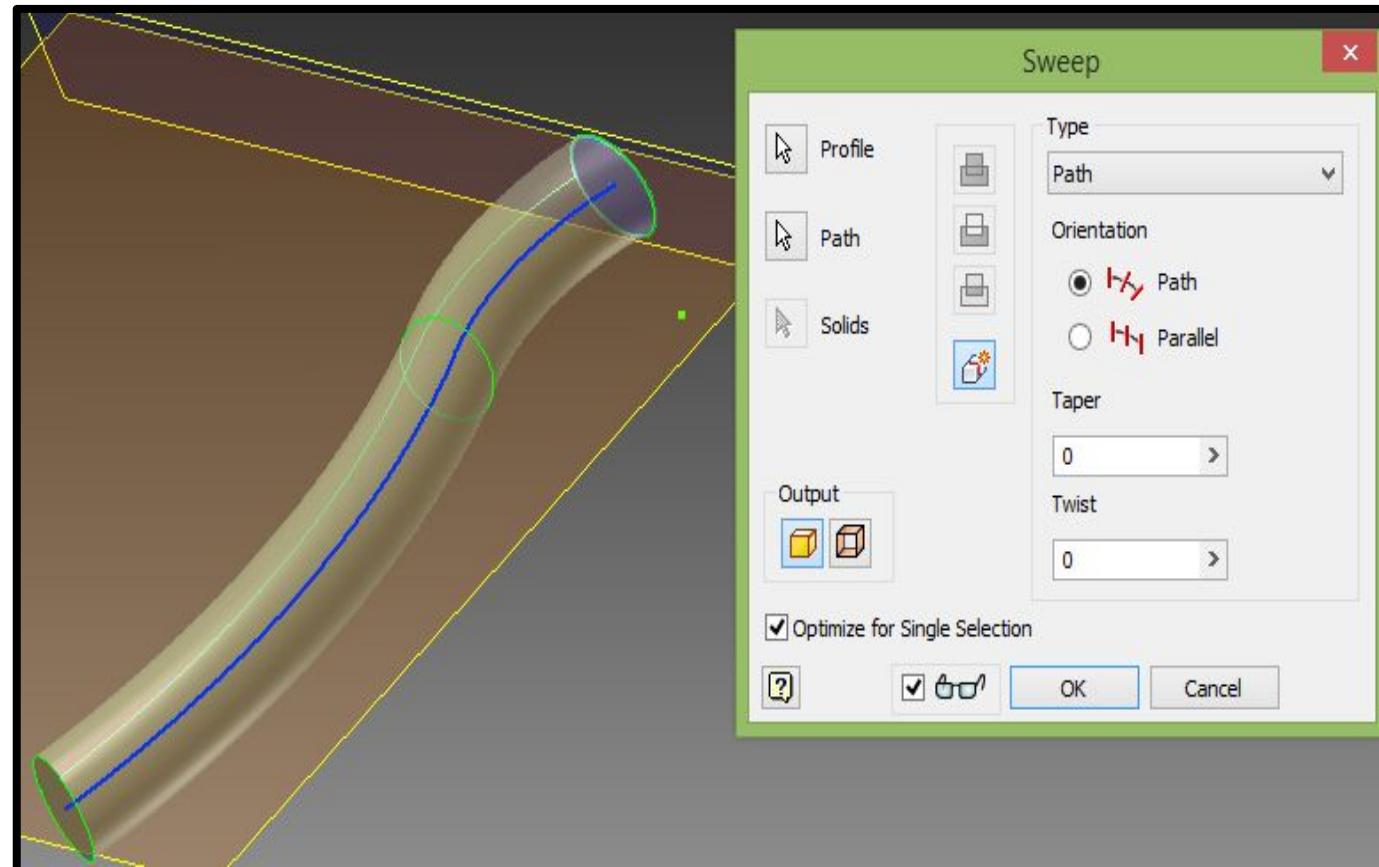
LOFT

- The loft technique chains together **multiple** profiles in a **linear** fashion. It can be useful for modelling parts of varying cross-sectional areas (e.g. ducts or nozzles).
- Lofts require:*
 - At least one custom work plane*
 - At least two profiles*
- Let's demonstrate this by lofting a rectangle and a circle profile.
- As a challenge, chain together five profiles each with a different cross-section.



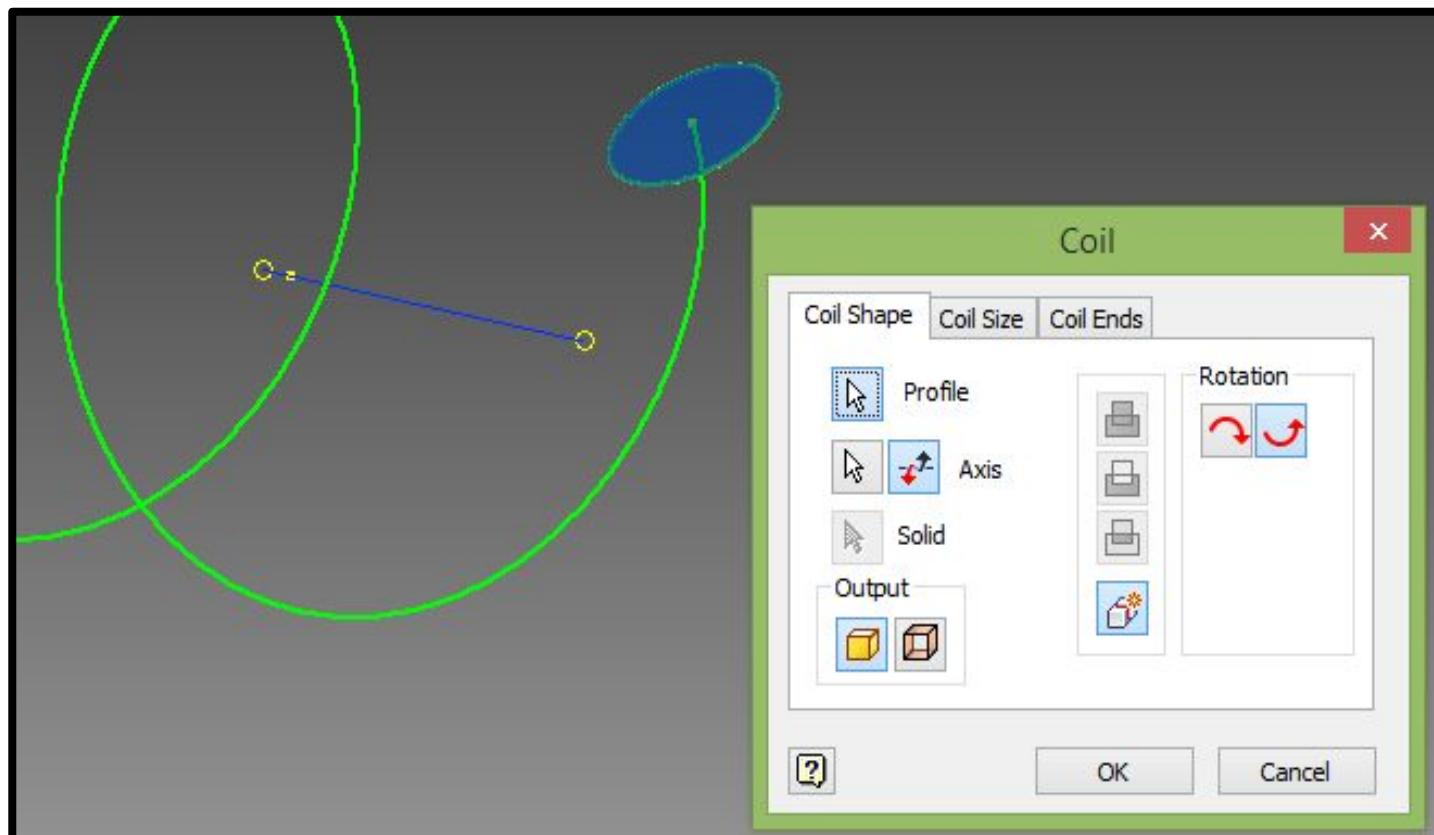
SWEEEP

- The sweep method extrudes a cross-sectional profile over a **non-linear path**. (i.e. pipes that would have bends in them)
- Sweep requires:**
 - One cross-sectional profile*
 - One perpendicular plane*
 - One path line that intersects the origin of the cross-sectional profile*
- Let's demonstrate this by creating a non-linear cylinder shaped solid.
- Take note that, sweeps with bends that are too **sharp** will have problems generating.



COIL

- The coil technique will sweep a cross-sectional profile about a self-generating helical path line. (e.g. a mechanical spring)
- Coil requires:*
 - One cross-sectional profile*
 - One reference axis*
- Let's demonstrate this by creating a spring.
- Take note of the **helical path generation options**, and the **taper options**.

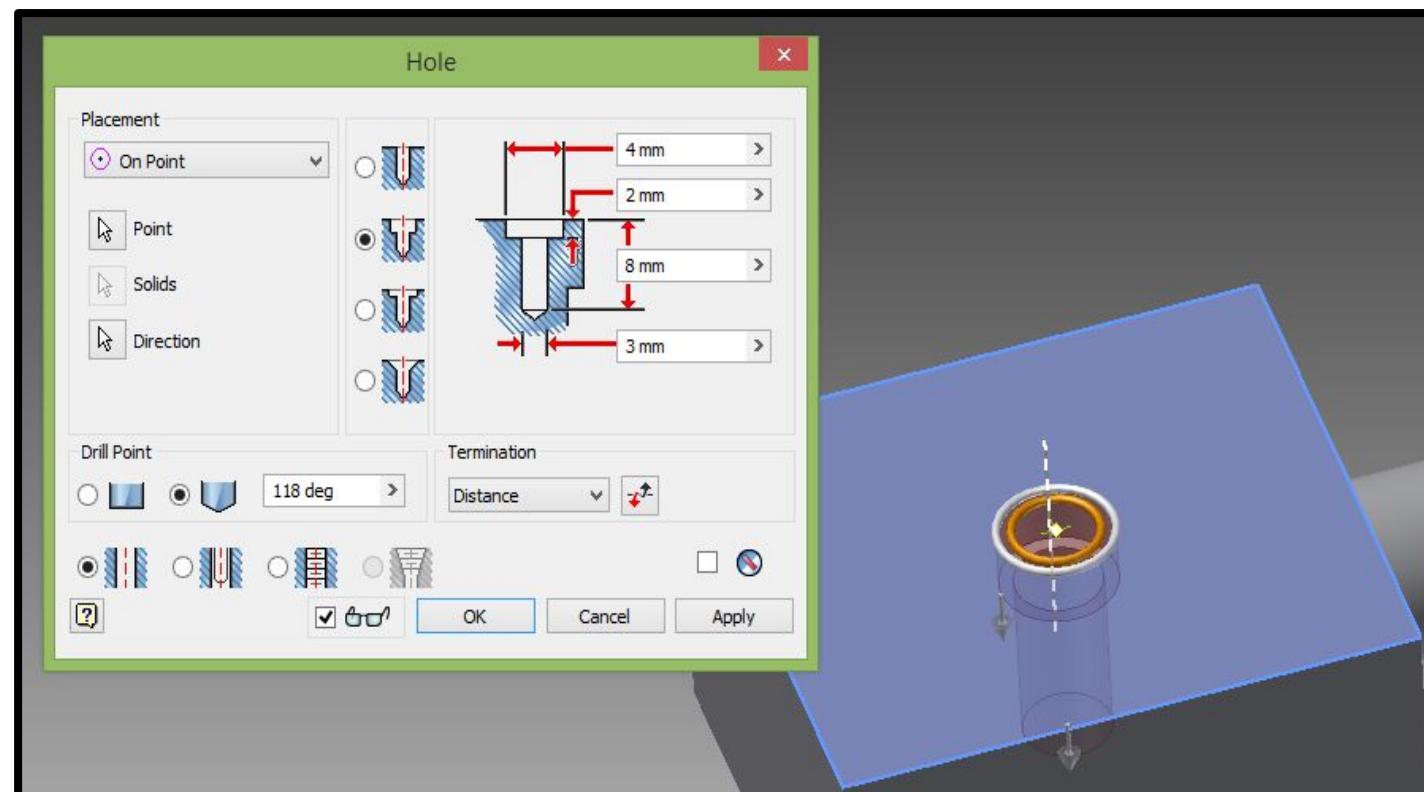


MODIFICATION FEATURES

- Modification features help to shape solids that cannot be done or harder to model using the 3D modelling features.
- The types of modifications are: **fillet/chamfer, hole, and shell**.
- We have already seen the **fillet** tool in action. We can use the **fillet/chamfer** modification option to smooth out or blunt edges.

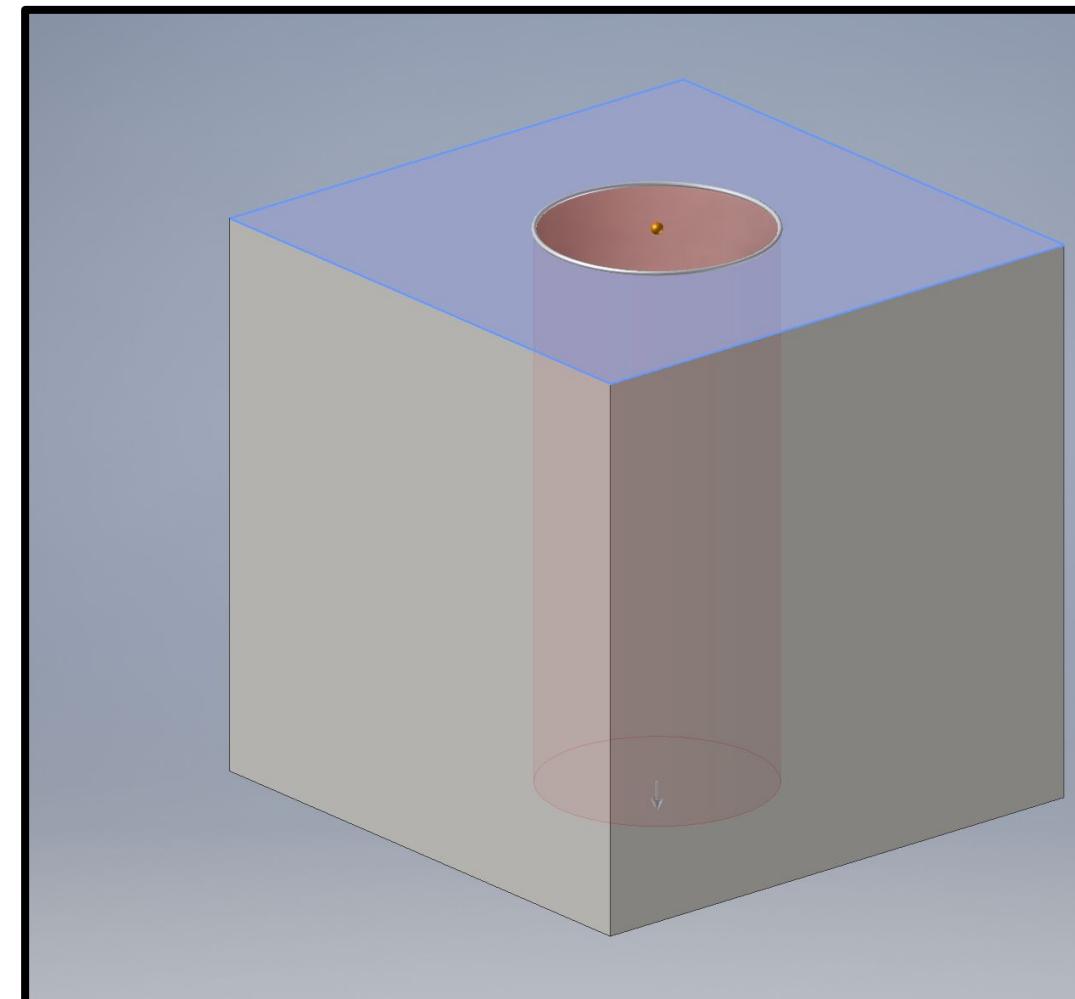
HOLE MODIFICATION

- Holes can be placed on surfaces, they type of placement needed will depend on the surface you are using.
- We did something similar with the secondary extrusion exercise, but hole modifications provide **more variety** and are capable of creating **more complex hollow sections**.
- Let's do an demonstration of placing a hole through the centre of a cylindrical solid. *Using the concentric placement method.*
- Take note of the hole types, cross-section dimensioning options, drill point.



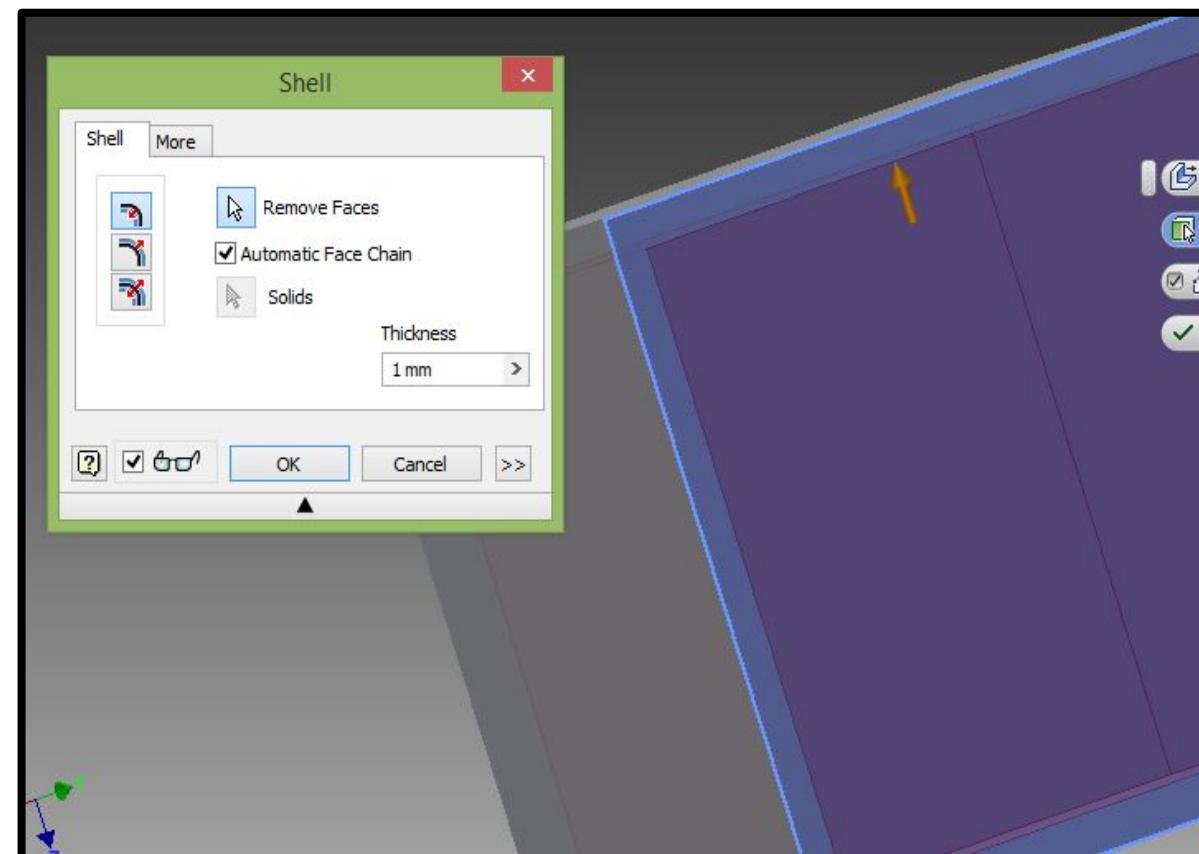
HOLE CHALLENGE

- Going back to the secondary extrusion exercise, try to recreate it in a different way.
- Use the hole modification to place a hollow section in the centre of a cube solid.
 - You will need to use the linear placement method.
- *When you finish, can you put in a second hole that could fit a bolt or screw?*

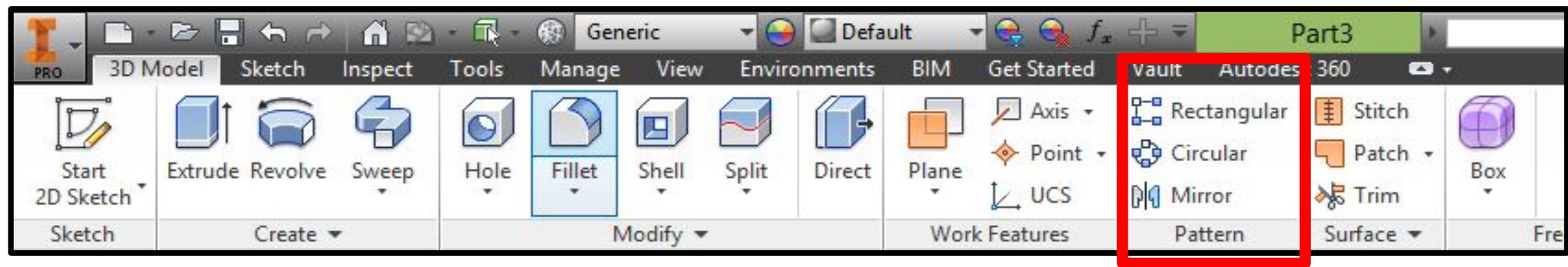


SHELL MODIFICATION

- The shell modification **hollows out** solid shapes and can even **remove faces**.
- The thickness of the walls can be specified.
- Before we suggested that sweep solids can be used to model pipes. Let's demonstrate how we can combine shell with sweep to achieve this.
- We also suggested that loft solids can be used to model nozzles, use the shell modification on the loft you made.



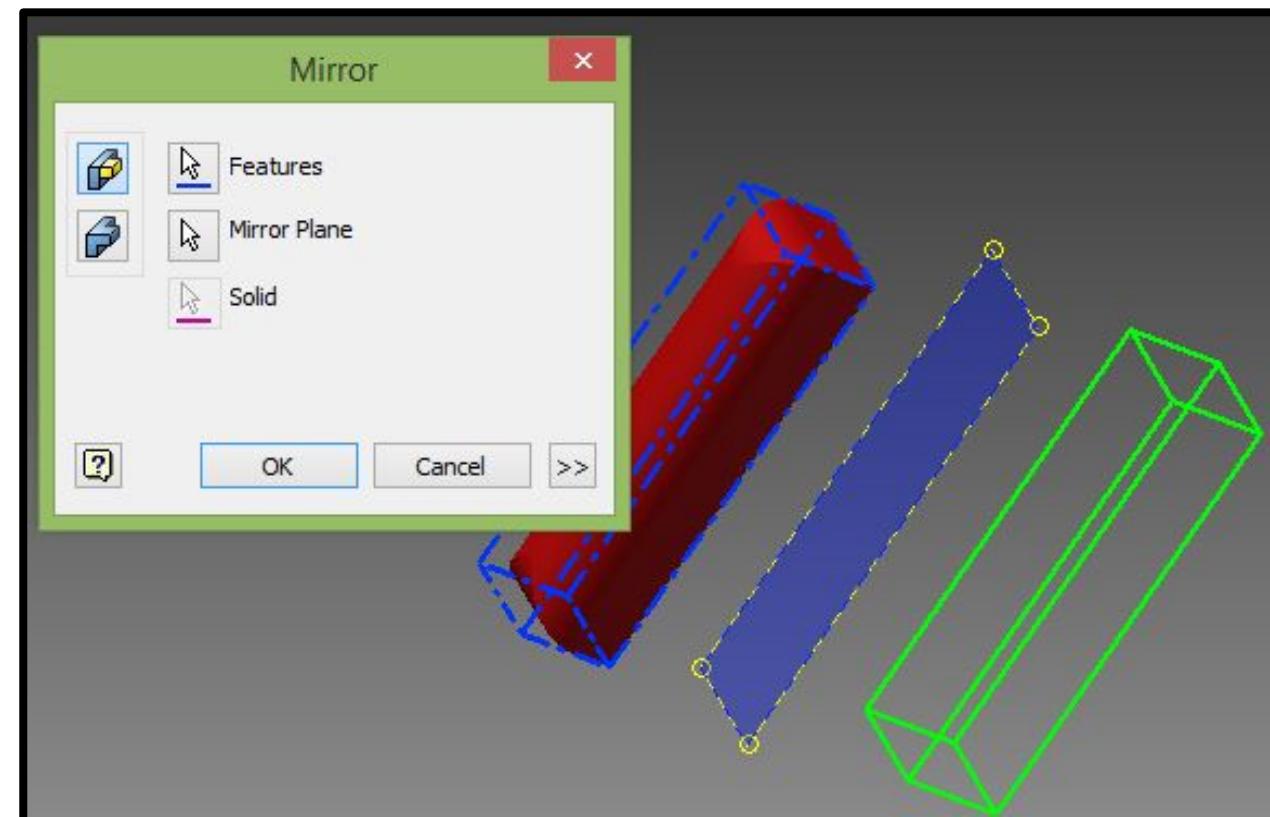
PATTERNING



- In more complex designs, repeating features can be tedious especially when there are many of them.
- With the patterning modification options we can repeat these features using a single operation to make this process more efficient.
- The types of pattern methods are: mirror, rectangular pattern and circular pattern.

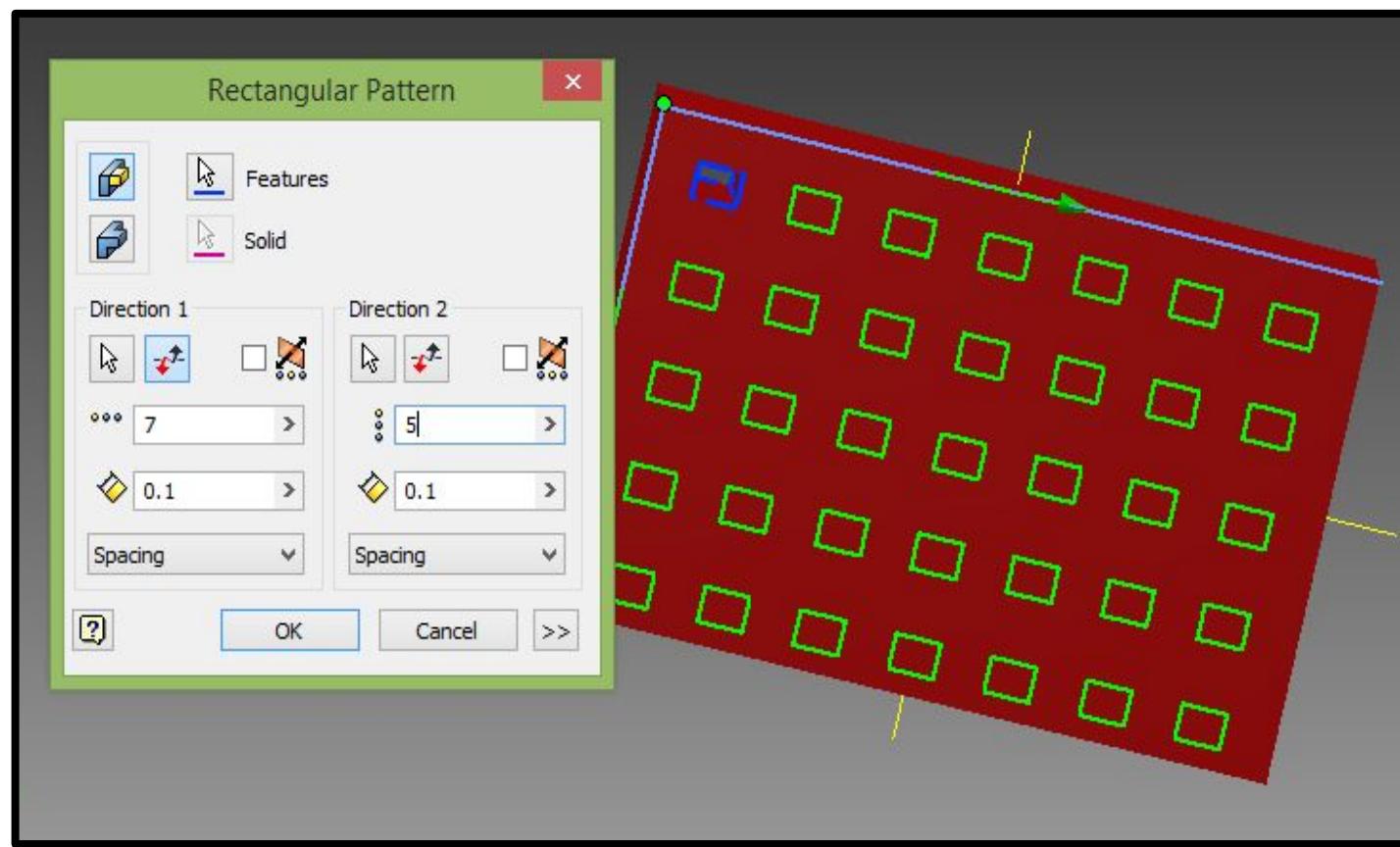
MIRROR PATTERN

- Mirror pattern will **reflect** a solid shape about a reference plane.
- *Mirror requires:*
 - At least one solid feature
 - One plane
- Let's demonstrate by mirroring any of our previous solids.
- Take note that more than one feature can be reflected at once. Hold 'shift' and select them from your model browser to make it easier.



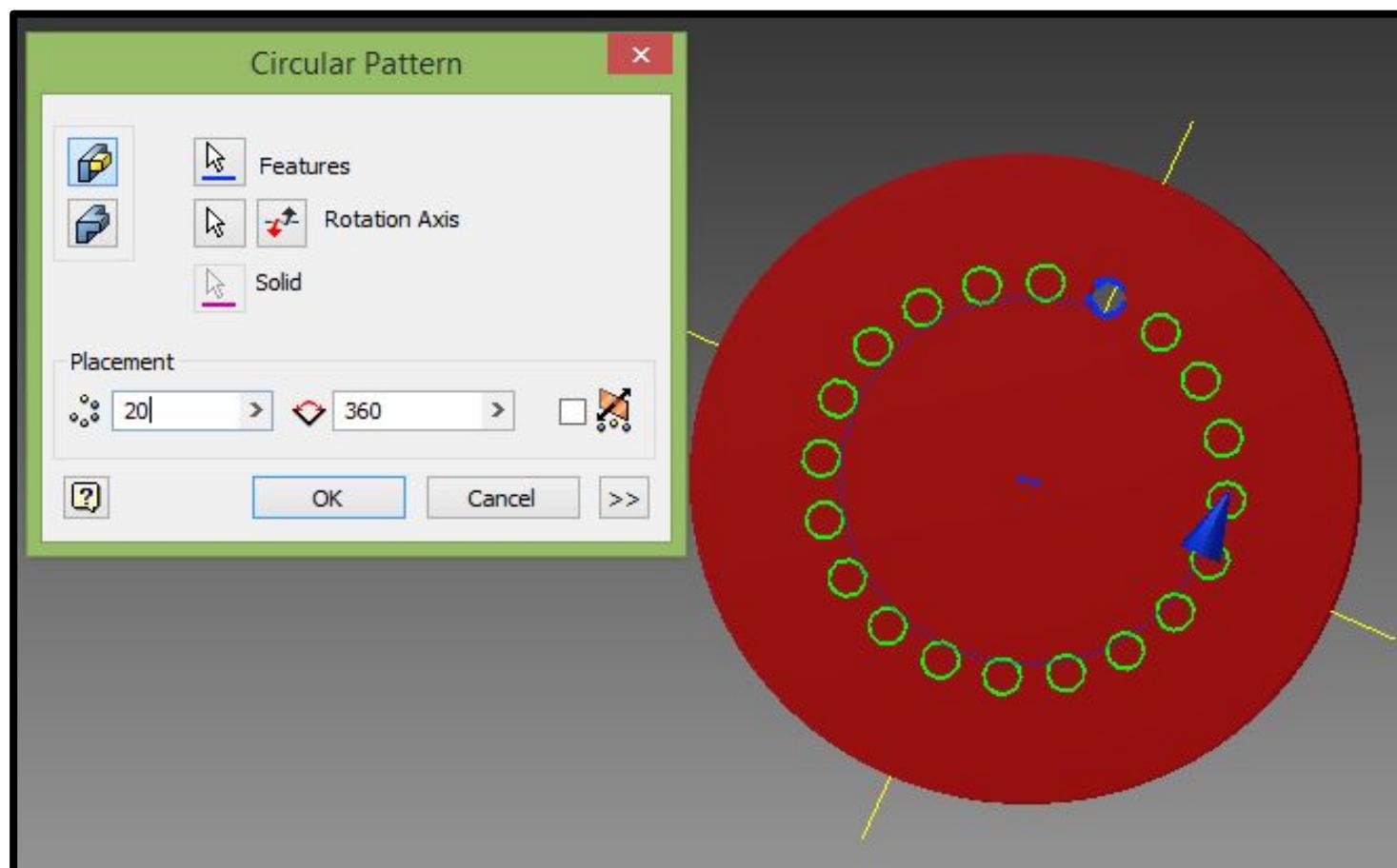
RECTANGULAR PATTERN

- Rectangular pattern repeats features over certain intervals in the horizontal and vertical (linear) directions.
- Rectangular pattern needs:
 - At least one feature
 - At least one reference edge
- Let's demonstrate this as in the example shown.



CIRCULAR PATTERN

- Circular pattern repeats features over certain intervals (measured in degrees) equidistant about a curved distance.
- Circular pattern needs:
 - At least one feature
 - One reference axis
- Let's demonstrate this as in the example shown.



ANY COMMENTS ABOUT THE COURSE? ~5MIN BREAK

- Let's take a 5min break to do a quick survey about the course so far.
 - [AutoDesk \(Drawing and Printing 3D Objects\)](#)
- Please fill out the form below:
 - <https://goo.gl/forms/bxkzXXKihs1PRrhx1>
- It's completely anonymous, any feedback really helps us to create better content for you all!
- Here's a table for what the number mean:

Score	Score Conversion (according to test...)	What that means...?
1 - 6	Detractor (Negative Reaction)	I thought the workshop was a waste of time. I would tell people not to go to this workshop.
7 - 8	Neutral (Passive)	I did not find the workshop that useful (didn't learn anything new). I would not say anything good or bad about it.
9 - 10	Promoter (Positive Reaction)	I found the workshop helpful and learnt something new. I would tell people to go to this workshop.



MODELLING EXAMPLE, VISUALISATION AND STL CONVERSION

LESSON 5

MOTIVATION

- So far we have looked at all the tools you will need to start modelling your own designs.
- But you still might be wondering in what situations can I use these? Or how can I combine these tools to actually model my design.
- In this lesson we will be getting you to apply these skills and see how they can be used to great design effect using the example of a fan head component.
- Your task is to recreate the part based on the images and information that you have been provided.
- Solutions provided:
https://docs.google.com/a/student.unimelb.edu.au/presentation/d/1_m2DTejcrWj1rG5JT4rn7P63hKxTU43GekQ1h44tRPA/edit?usp=sharing



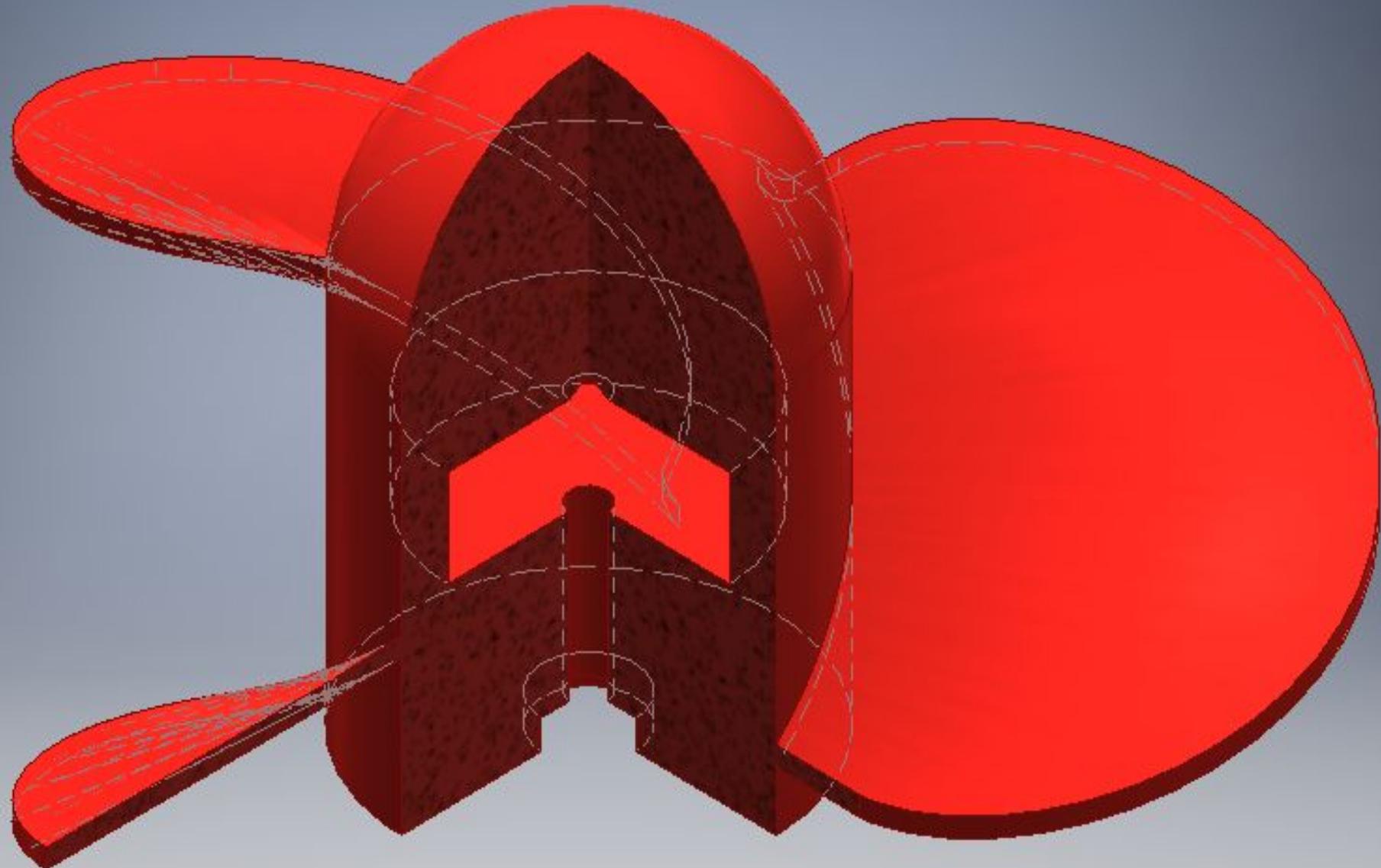
Model

Inventor2016_FanExample

- Solid Bodies(1)
 - View: Master
 - Master (checked)
 - ThreeQuarter
- Origin
 - YZ Plane
 - XZ Plane
 - XY Plane
 - X Axis
 - Y Axis
 - Z Axis
 - Center Point
- Extrusion1
- Revolution1
- Work Plane1
- Extrusion2
- Hole1
- Coil1
- Fillet1
- Fillet2
- Circular Pattern1
- End of Part

The main workspace displays a 3D model of a fan blade, colored red. The model consists of two main circular components and a central hub. A coordinate system (X, Y, Z) is visible at the bottom left. To the right, a navigation cube shows 'FRONT', 'TOP', and 'RIGHT' views. The bottom navigation bar includes icons for file operations (New, Open, Save, Print, etc.) and tabs for 'My Home' and 'Inventor2016_FanExample.ipt'.

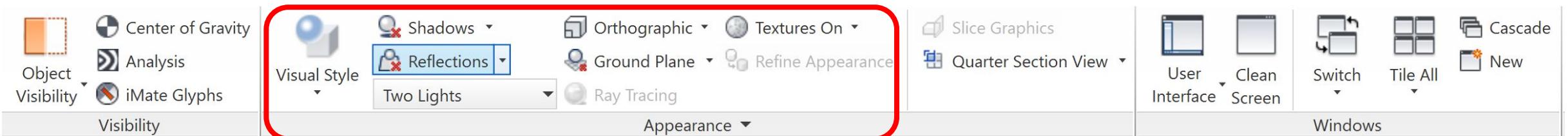
Ready



FAN EXAMPLE SPECIFICATIONS

- The inner cylinder has a circular profile centred at the origin, diameter = 200mm and extruded height = 200mm.
- The cylinder nose is a hemispherical end with radius = 100mm.
- The internal chamber has a circular profile centred at the origin, diameter = 150mm. The height of the internal chamber is between 100mm and 150mm from the base of the part.
- There is a “counterbored” hole centred on the bottom surface of the part. The counterbore diameter = 50mm, counterbore depth = 15mm, hole depth = 150mm, hole diameter = 20mm.
- The fan blade has a 10mmx200mm rectangular profile, beginning 35mm from the base of the part.
- The blade has a revolution = 0.25ul and height = 150mm.
- The curvature of the outer blade ends = 120mm and the inner shoulders of the blade have a curvature = 5mm.
- There are 3 occurrences of the fan blade placed equidistant about the inner cylinder.
- The part can be viewed absolutely and with a three-quarter sectional view revealing the cut out sections for the internal chamber and shaft.
- The visual style of the part is ‘shaded with hidden edges’ as to emphasise the internal construction of the part.
- The chosen material for the part is ‘LCP Plastic’ and has an ‘orange-red’ outward appearance.

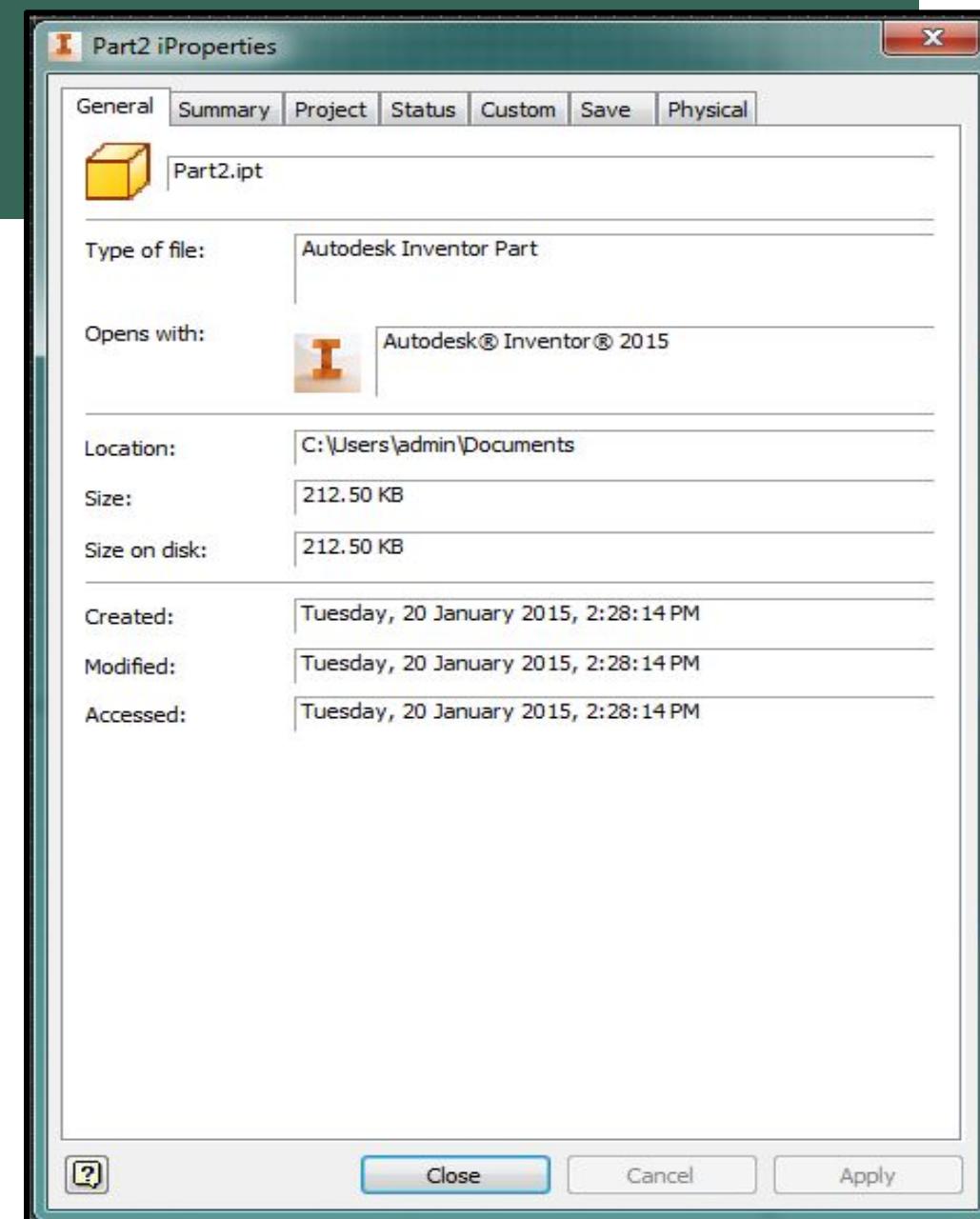
VISUALISATION TECHNIQUES



- In our example, it might not be clear to all viewers that there are internal features.
- Visualisation techniques can enable you to present your component such that important features can be clearly identified.
- You can select visual styles that have **hidden lines** presented.
- You can create different viewing perspectives, i.e. sectional views.

IPROPERTIES

- iProperties are a set of information that you can define for the part file, including: general file settings, project relevance, custom notes, etc.
- More importantly, the iProperties enable **material selection** and can even calculate the inertial properties for the component based on its material.
- The iProperties menu can be found through the main Inventor icon (top-left), right-clicking on the component or by right-clicking on the component in the model browser.
- **Material textures** can be changed by accessing the appearance and colour palettes. *Located in the top ribbon.*

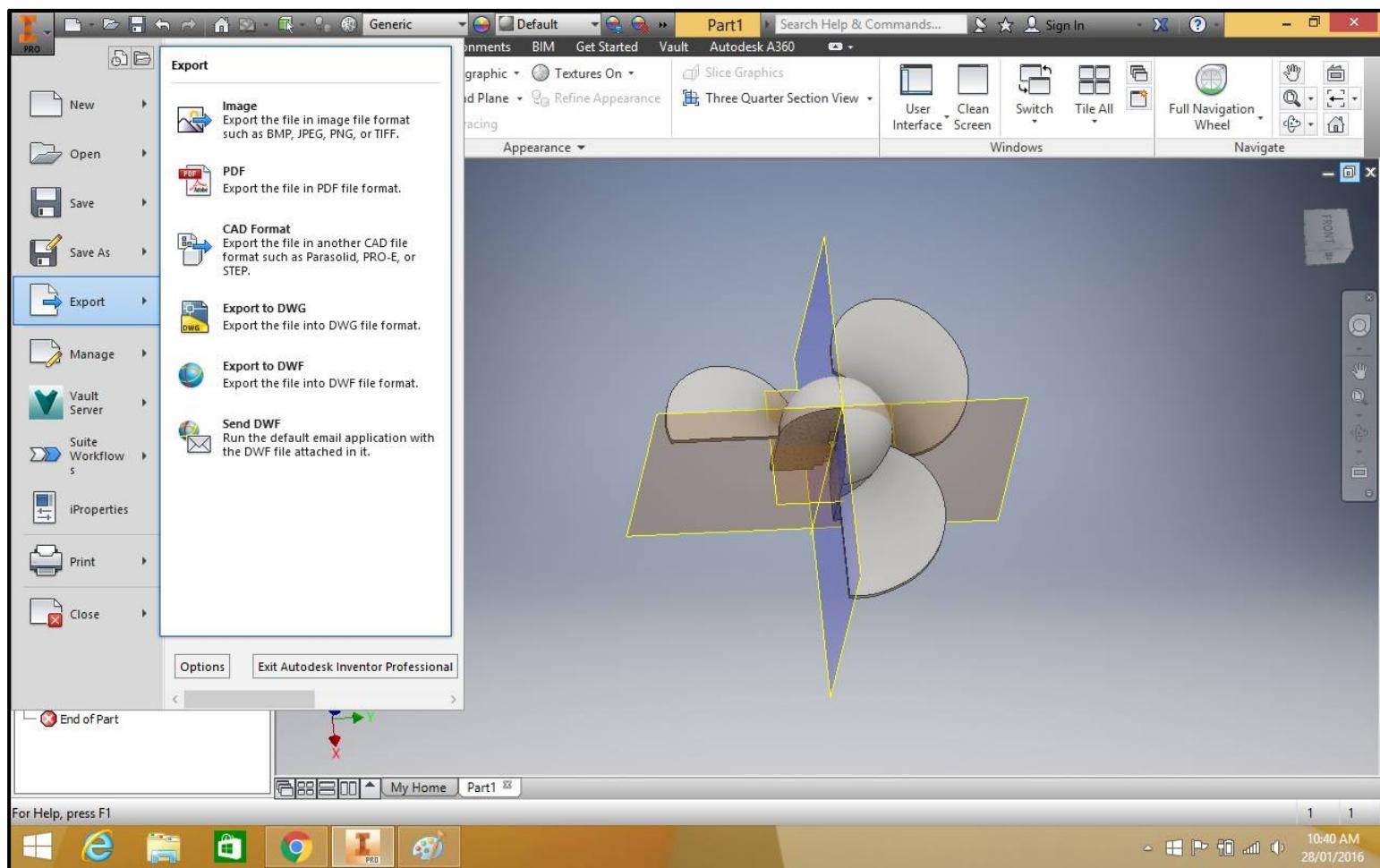


STL CONVERSION

- What happens after we have completed our design and want to fabricate it?
- One available option is to **3D Print** our component.
- To get the part file (**.ipt**) into a format that can be read by a printer software we must convert into an **STL** format (**.stl**) or (**.step**).
- **.stl** is probably the more common format, but when printing always be sure to check first what format is acceptable for that particular printer.

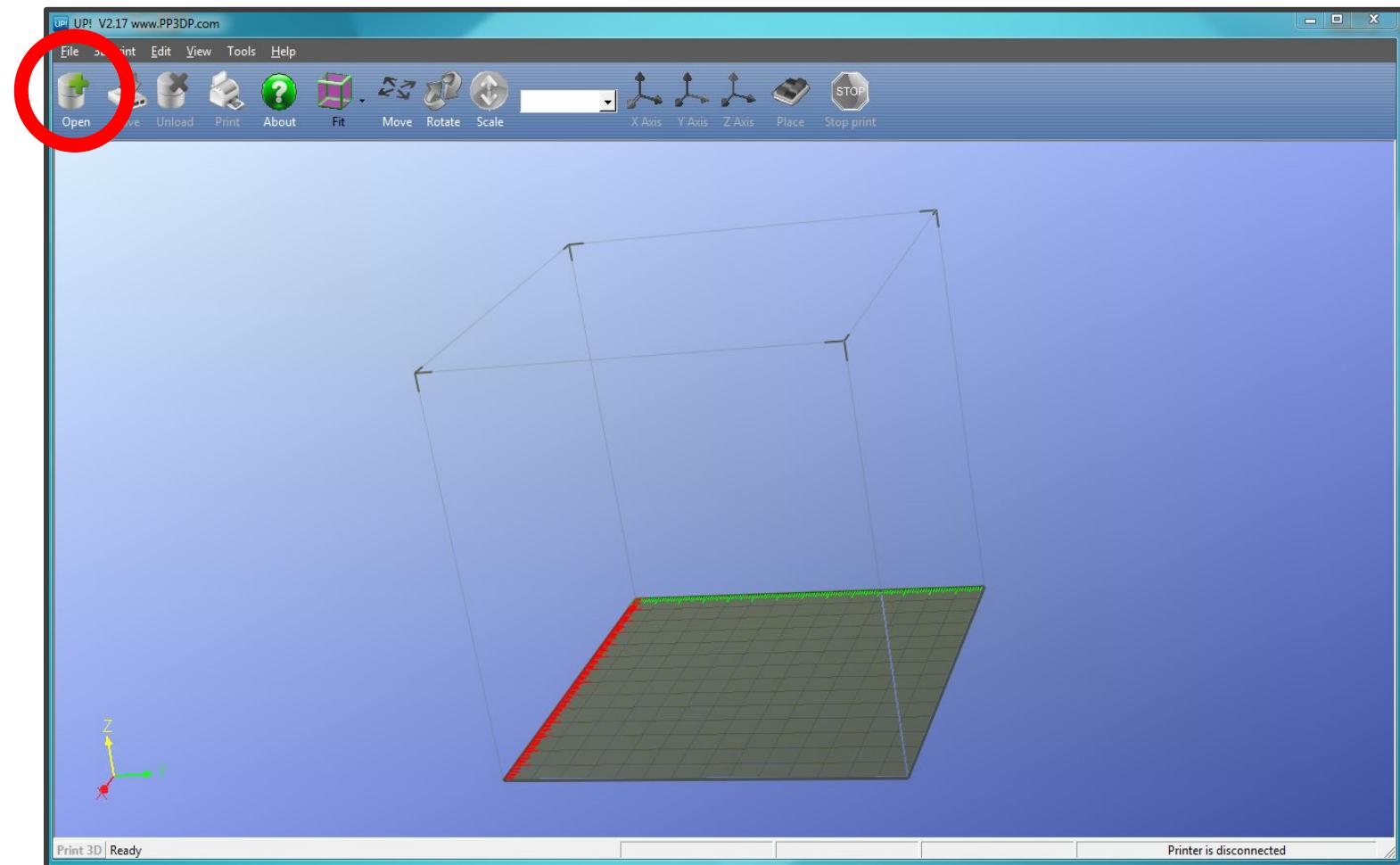
EXPORT TO STL FORMAT

- The export to STL format option is found under the Inventor icon.
- Select Export > CAD Format
- When you ‘save as’, change the extension to a .stl or .step depending on what you need.



3D PRINTING

- In your own 3D printing software application, you will be able to import your designs as STL files.
- Place and orientate your component for printing.
- Ensure you have your printer properly calibrated and initialised.
- *Your interface might look differently to what I have here.*



CONCLUSION

- You have used everything you have learnt today to 3D model a single component.
WELL DONE!!
- The iProperties can be used for material selection and computing physical properties.
- We looked at STL conversion so that you can 3D print/fabricate and physically realise your creations.
- These could have implications on the simulation and manufacturability of components of your design.



THANK YOU FOR ATTENDING!

If you are interested in more trainings or events: <http://melbourne.resbaz.edu.au>