



# AUTODESK INVENTOR COMPUTER-AIDED DESIGN QUICKSTART PROGRAM 2017

# DESIGNED FOR ENGINEERING STUDENTS



THE UNIVERSITY OF  
MELBOURNE

# THE QUICKSTART PROGRAM

- The QuickStart Program is a consultation session for computer-aided design problems accompanied by a formal teaching component using Autodesk Inventor 2016 software.
- The program will run between week three and eight (excluding week four) of semester 2, where will be two sessions held each week introducing new CAD concept: in Old Engineering Building, EDS-5.
- Session times are:
  - Days: Thursday & Friday (*these will be repeat sessions each week*)
  - Times: 5:15pm – 7:00pm (*student mentors will be available until 6:05pm*)
  - Place: Old Engineering Building, EDS-5
- Any questions outside of the consultation sessions can be emailed in:
  - Email: [mcenCAD17@eng.unimelb.edu.au](mailto:mcenCAD17@eng.unimelb.edu.au)

# PROGRAM SCHEDULE

- Each of the QuickStart sessions will have (at least) two mentors present. And will begin with a 30min demonstration of a basic CAD concept, the remainder of the session will be left as a general consultation for asking questions or to work on other projects.
- The teaching schedule for the program is as follows:
  - Week 3: Parts Modelling
  - Week 5: Assembly Modelling
  - Week 6: Motion Simulation & Exploded Presentations
  - Week 7: Technical Drawings
  - Week 8: Stress Analysis (*in Inventor*)



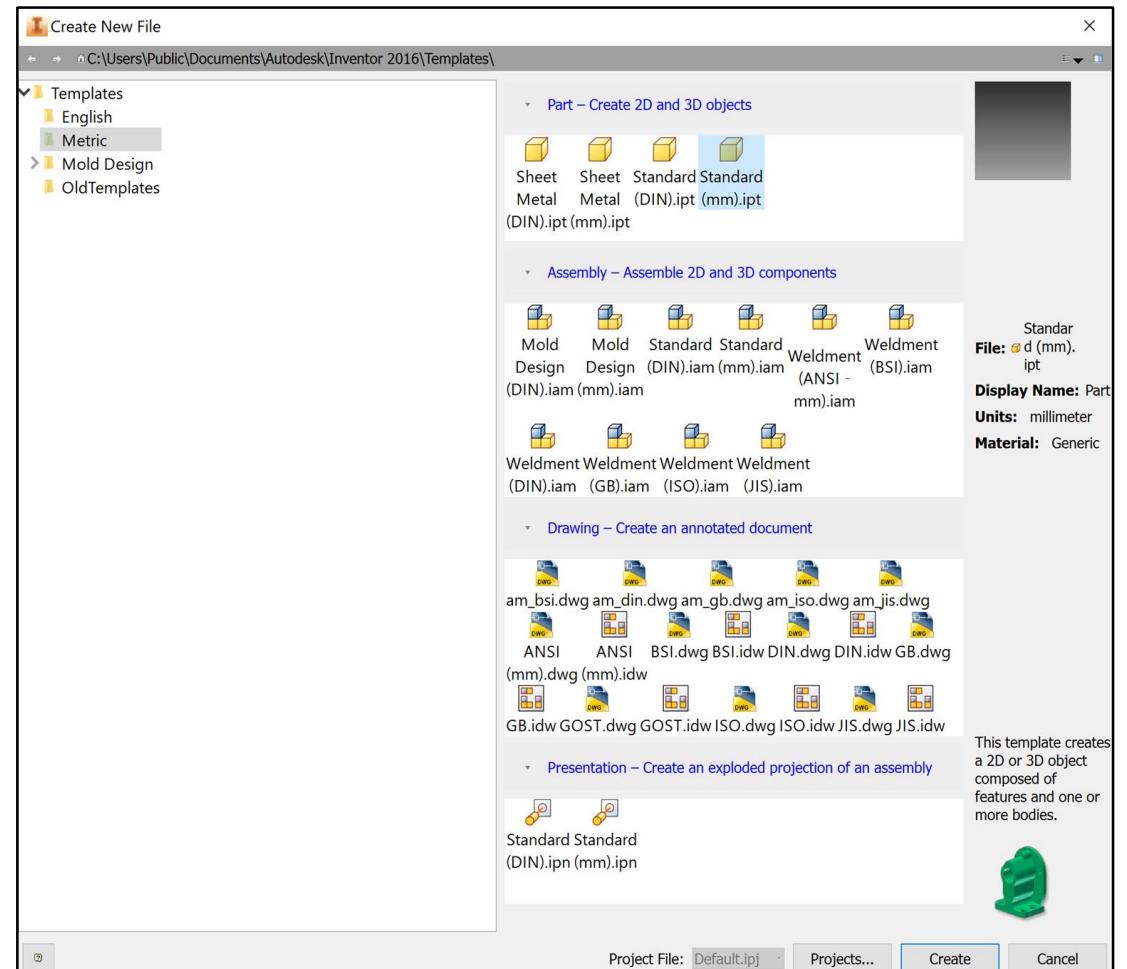
# THE BASICS

***LESSON 0***



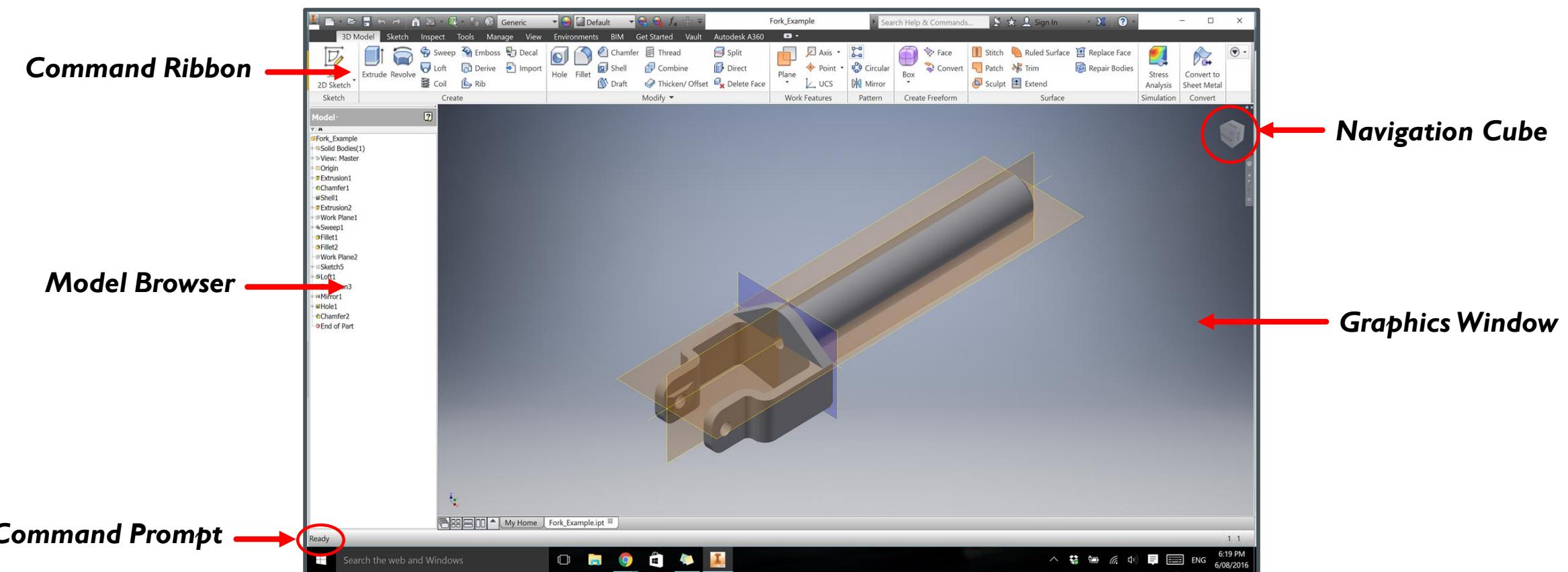
# ABOUT THE PROGRAM

- Inventor is a computer-aided design software developed by Autodesk used for product design, rendering and simulation.
- The program itself consists of four different modules, or design environments:
  - Parts | Assemblies | Presentation | Drawings**
- Each of these are designed to assist with visualising engineering or product design at various stages of the design process.
- When creating a new file, these modules are differentiated by their file types.



# PROGRAM INTERFACE

- Open the 'fork\_example' file in the lesson materials.

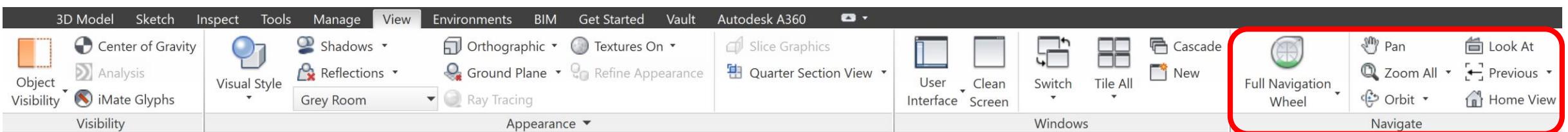


# THEVIEW CUBE

- The **view cube** determines how the graphics window is orientated, or in other words the user's viewing perspective.
- The cube is also related to the xyz coordinate system of the workspace (*the xyz axes can be located in the bottom-left of the graphics window*), such that selecting a face, edge or corner of the view cube will show a different perspective by rotating the coordinates.
- A default **home** view can be set such that the view (perspective) can be returned to at any point.
  - To set a default view, *right-click on the house icon to the left of the view cube, and selecting the 'set current view as home' option.*



# NAVIGATION COMMANDS



- There are three modes of movement in the 3D environment:
  - **Scale (zoom) | Translation | Rotation**
- You can either select the commands found in the navigation section of the command ribbon, use keyboard shortcuts, or use mouse controlled commands (**recommended**).
- Shortcuts:
  - **Scale (zoom): Hold F3 or Scroll with Mouse Wheel**
  - **Translation: Hold F2 or Drag with Mouse Wheel**
  - **Rotation: Hold F4 or Shift + Mouse Wheel (Drag)**



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# THE MODEL BROWSER

- The model browser is a **chronological list of items** that are currently in the workspace. Each item is a component in the 3D model and their order can be changed (*moved up and down the list*), however some are dependant on preceding components thus changing their order of occurrence is not always possible due to the **parent-child dependencies**.
- The parent-child dependency can also be exemplified by the sketch-component relationship. Expand any 3D component and you will find that there is a 2D sketch underneath it, thus the sketch is required for that component to exist.
- It is possible to select and/or edit components straight from the model browser. Right-click an item in the browser to its menu, notice options for edit, delete and **suppress**.
  - Suppressing a feature will remove it from existence in the workspace, such that un-supressing it will return it to the design.
  - The '**end of part**' item in the browser serves a similar purpose, all items after it will be suppressed.
- Note that the first item in the model browser is the **origin**. This folder contains all the work features that are anchored to the origin point in the workspace. The **visibility** of these features can be toggled on and off, right-click on an origin item, and select the visibility option.



# PARTS MODELLING

***LESSON 1***

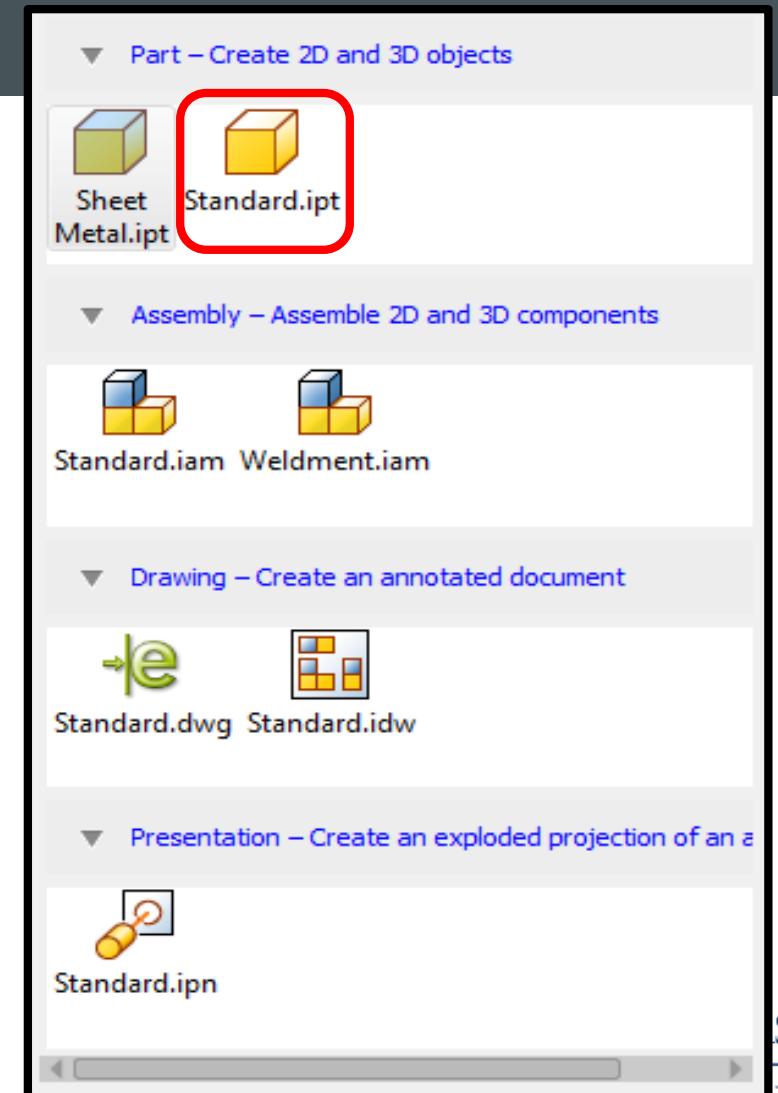


# PARTS MODELLING

- The part file is the most fundamental component of a CAD design and can be created through the part module where the *individual* modelling of components will occur.
  - Part files can come together to form a larger engineering design or simply act as a stand-alone model.
- In the ‘Parts Modelling’ section we will examine the two main processes involved in modelling a part component:
  - **2D Sketching | 3D Modelling**
- We will present a demonstration by modelling a part using an engineering design.
  - *All other examples are included for your own learning/practice.*

# CREATING A PART FILE

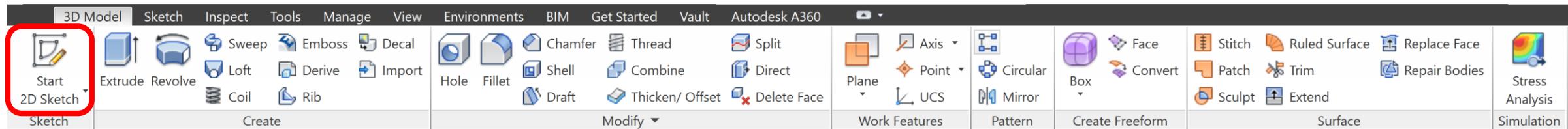
- A part file associated with the **.ipt** extension.
- The most basic type of part file is the **standard**, we will be using this throughout the rest of the workshop so always select this one when starting a new file.
- Most people tend to work in the **mm** units, but **inches** is also an available option.



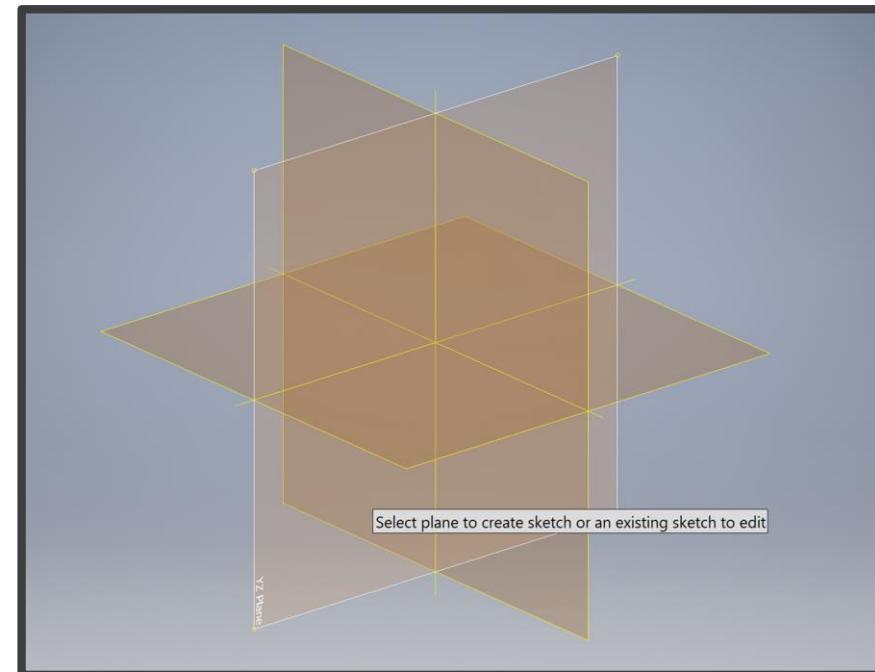
# 2D SKETCHING

- In Inventor 3D components dependant on sketches, or **profiles**.
- The general process for sketching a profile is:
  - Start a new 2D sketch.
  - Use drawing tools to create a sketch.
  - Apply modification tools to shape the sketch, if needed.
  - Apply constraints and dimensions to fully configure sketched elements (you should have one unique solution).
  - Finish Sketch.

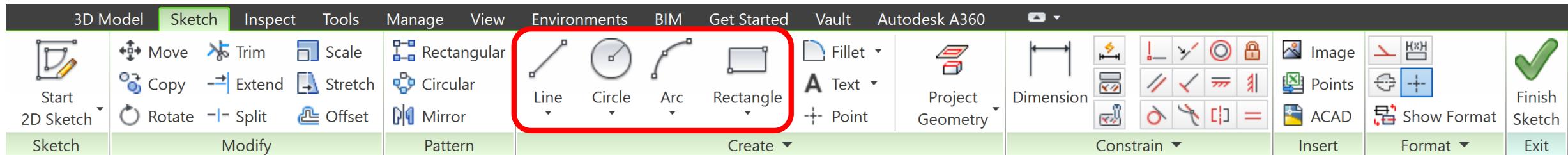
# STARTING A PROFILE



- Select the **start 2D sketch** command
- When creating a sketch for the first time the origin planes will appear, (select one to begin drawing on that surface).
- The program will move into a 2D drawing environment to commence the sketch.

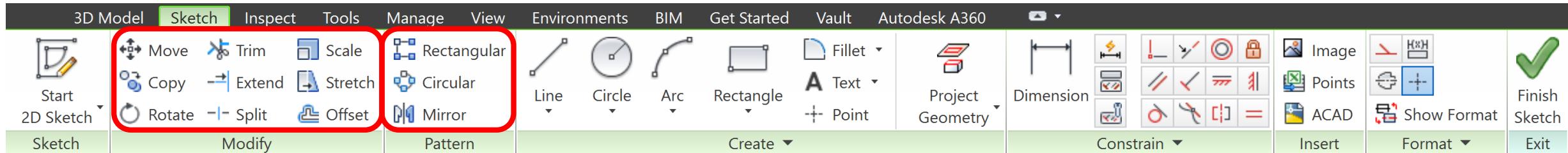


# DRAWING TOOLS



- Basic drawing types such as: line, circle, arc, rectangle and fillet, allow a range of drawing elements to be applied.
  - There are many different ways to draw these elements and in their drop-down menus variations exist such that a combination of them can form a more complex shape.
- The 'ctrl +' commands still apply i.e. copy, paste, undo.
- When unsure of how to execute an operation, hover over the option with your cursor to see a brief tutorial or refer to the command status (bottom-left).

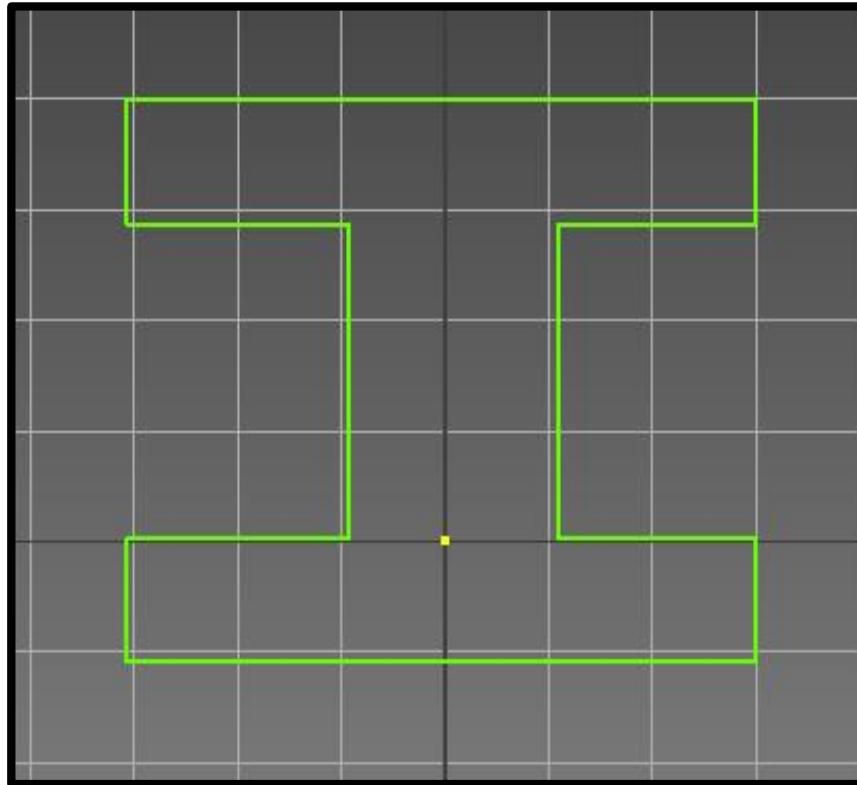
# MODIFICATION TOOLS



- Sketched elements can be **modified** if the desired shape cannot be drawn using just basic drawing elements, or for ease of manipulating drawing elements.
- Another important tip is never to draw or drag elements at random because this can lead to discontinuities in the sketch that can be difficult to notice at a glance.
  - Often zooming into a sketch can be helpful in ensuring that elements are in the correct place.
- The **patterning** tools can be used to repeat drawn elements.

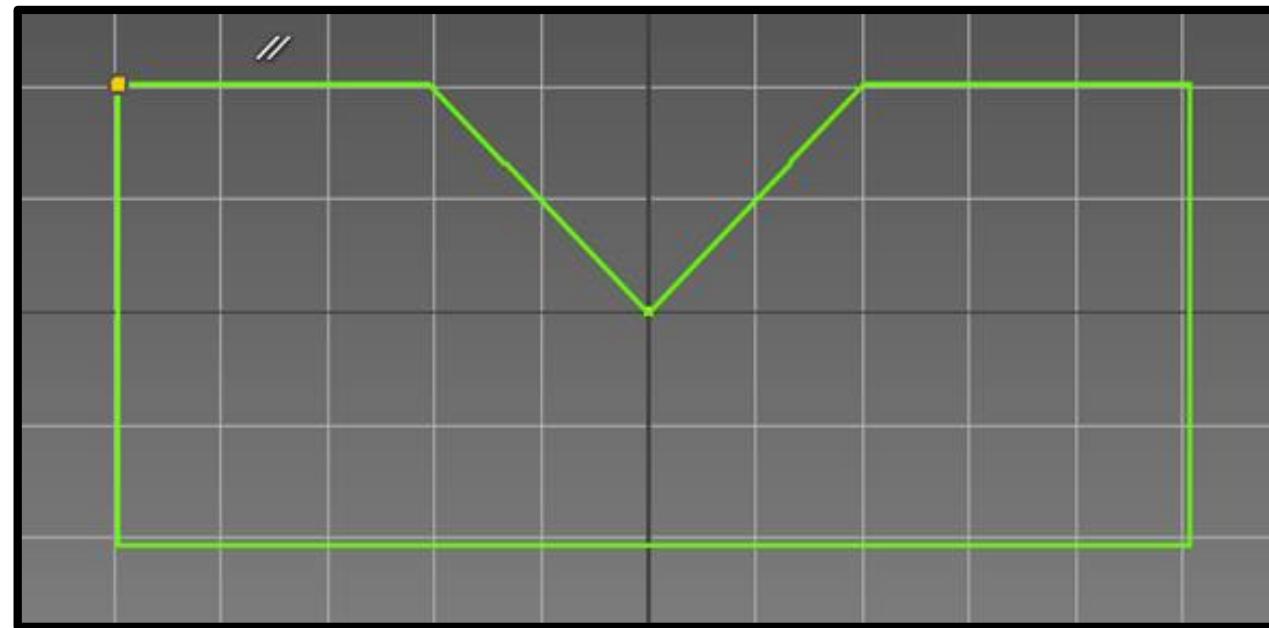
# PRACTICE EXAMPLE I: I-BEAM

- 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?

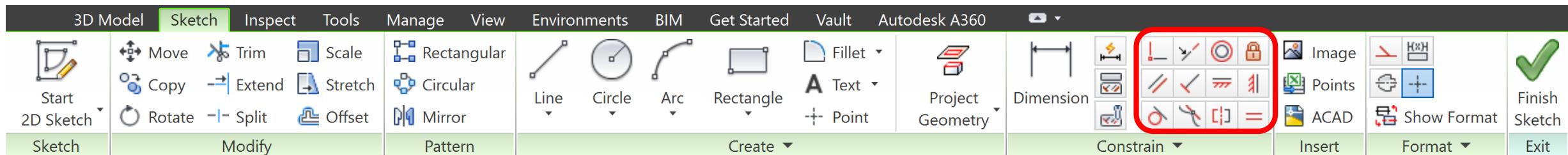


## PRACTICE EXAMPLE 2: NOTCH

- Draw the notch as shown. You just need to get a decent **shape** for now.
- There are a few ways to use both drawing and modification tools to create this, how do you want to approach this sketch?

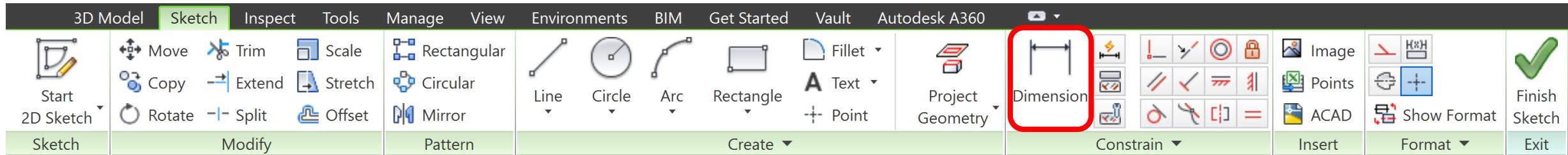


# CONSTRAINTS



- Constraints are applied to the sketch for two reasons:
  - Produce a **unique** solution for the profile
  - Fully constrain all drawn elements such that they cannot move
- Constraints are relationships imposed on specific elements such they are fixed in a certain orientation.
  - 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.
  - Hotkeys **F8** and **F9** will 'show all' and 'hide all' constraints respectively.

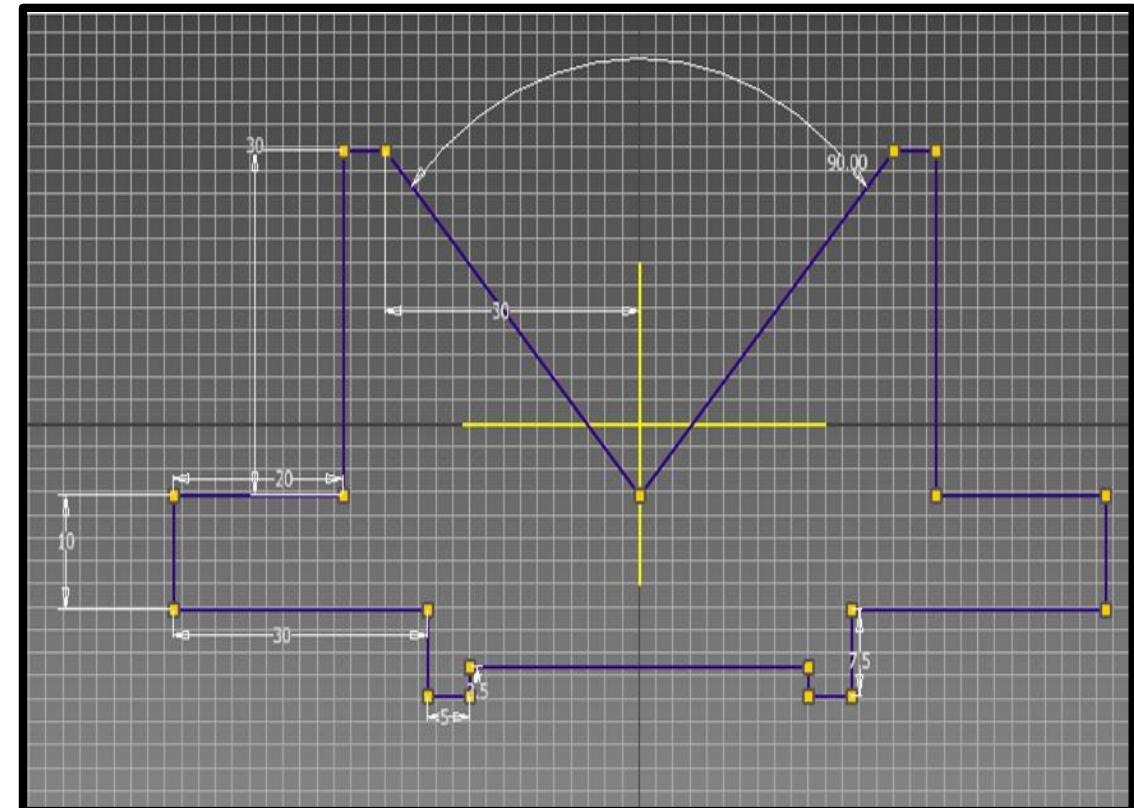
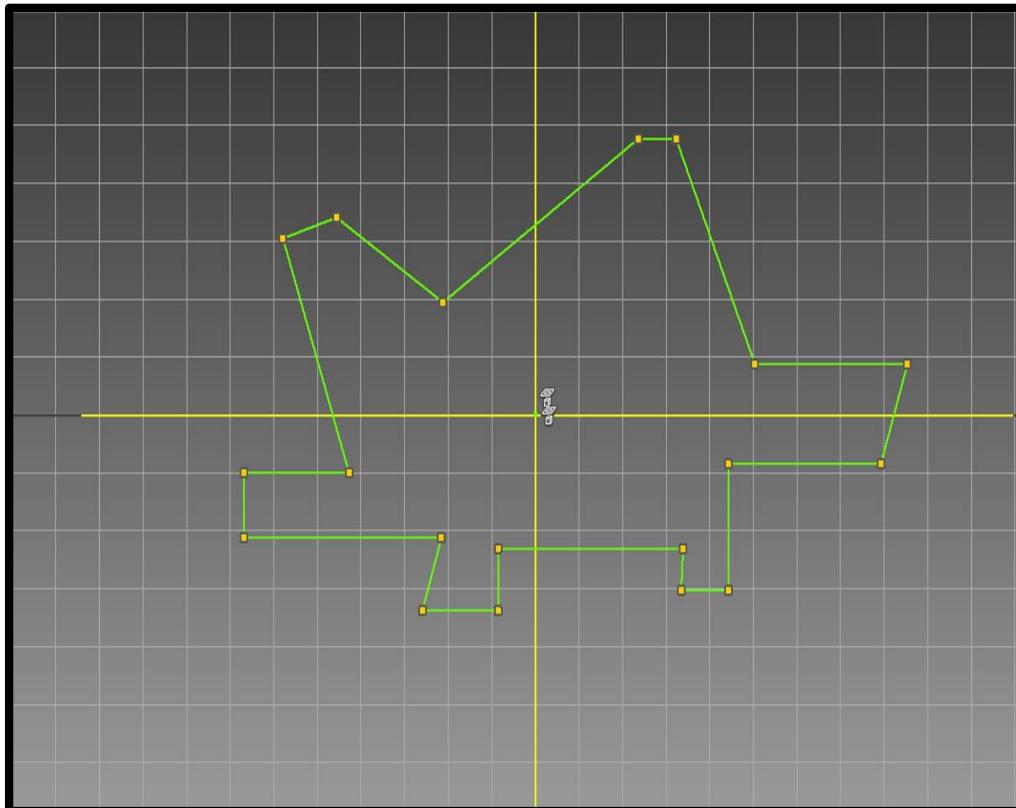
# DIMENSIONS



- A profile cannot be fully constrained without dimensions.
- Dimensions are (in a way) a type of the constraint technique because it, ultimately, restricts how the model appears, but more importantly dimensions provide people with information about the design.
  - They (dimensions) assign a numeric value for how long, wide, round a drawn element needs to be to complete the profile.
- *In the bottom-right corner a notification will indicate how many more dimensions are required to fully constrain the sketch.*

# PRACTICE EXAMPLE 3: CONSTRAIN & DIMENSION

- Open the **Exercise 3 Guide** and edit the **Exercise 3** file, found in the lesson materials.
- **HINT:** Use project geometry and symmetry constraint

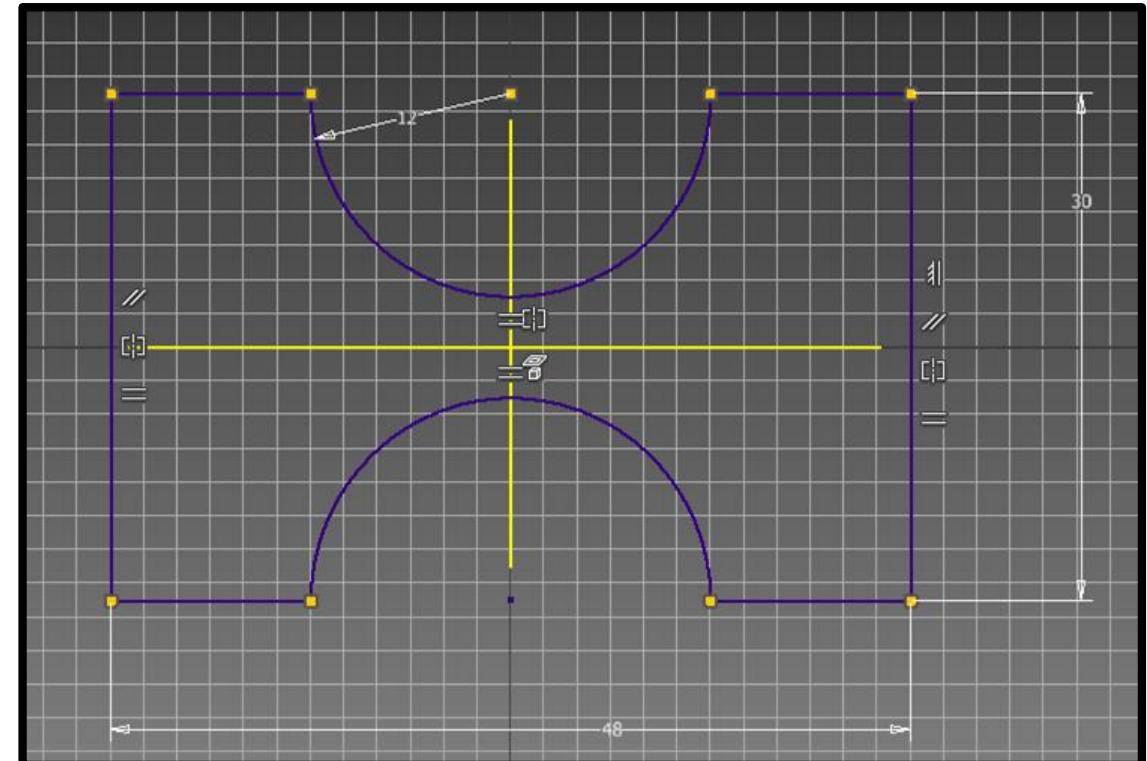
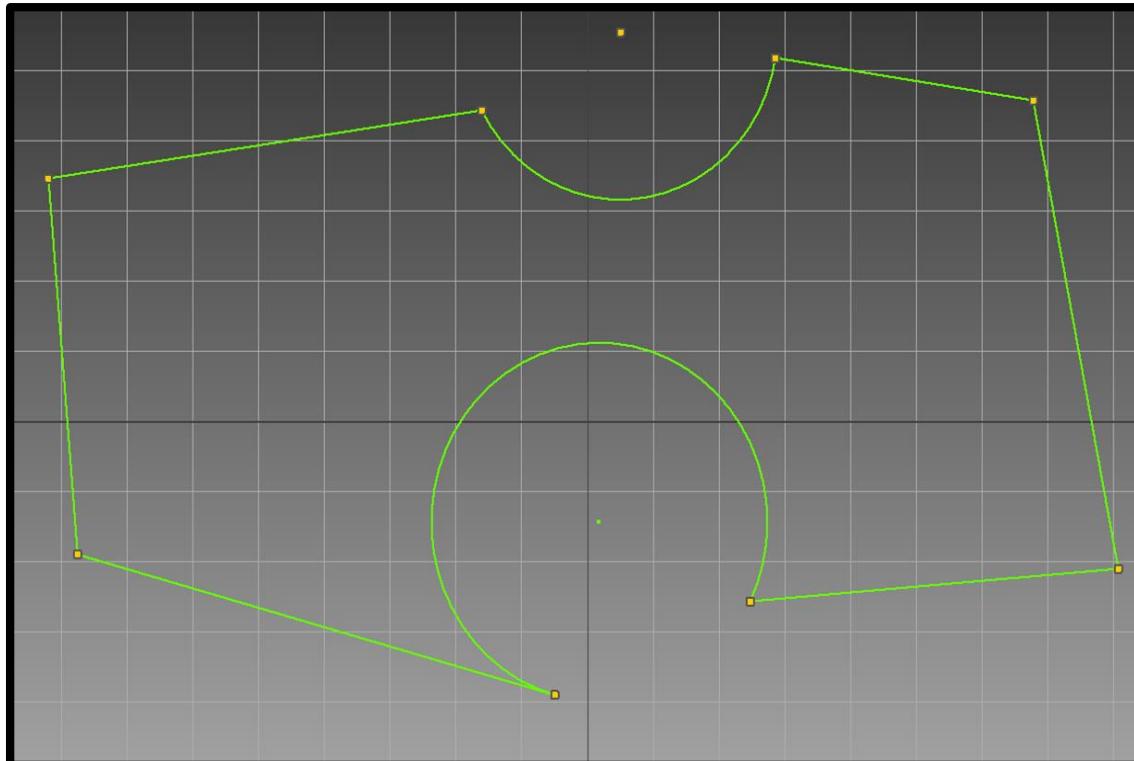


# PRACTICE EXAMPLE 3 REVIEW

- 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 constrain about the axis to halve your workload.
  - When constraining be willing to move around lines to maintain the general shape of the sketch.
  - While dimensioning, the sketch can be prone to doing strange things. Simply undo the operation and apply dimensions elsewhere so that the general shape is at least maintained.
  - Remember to also contain your sketch to the workspace (to the origin or the projected axes) so that it is completely fixed.
- *There is a strange dot in the centre of the sketch, delete it.*

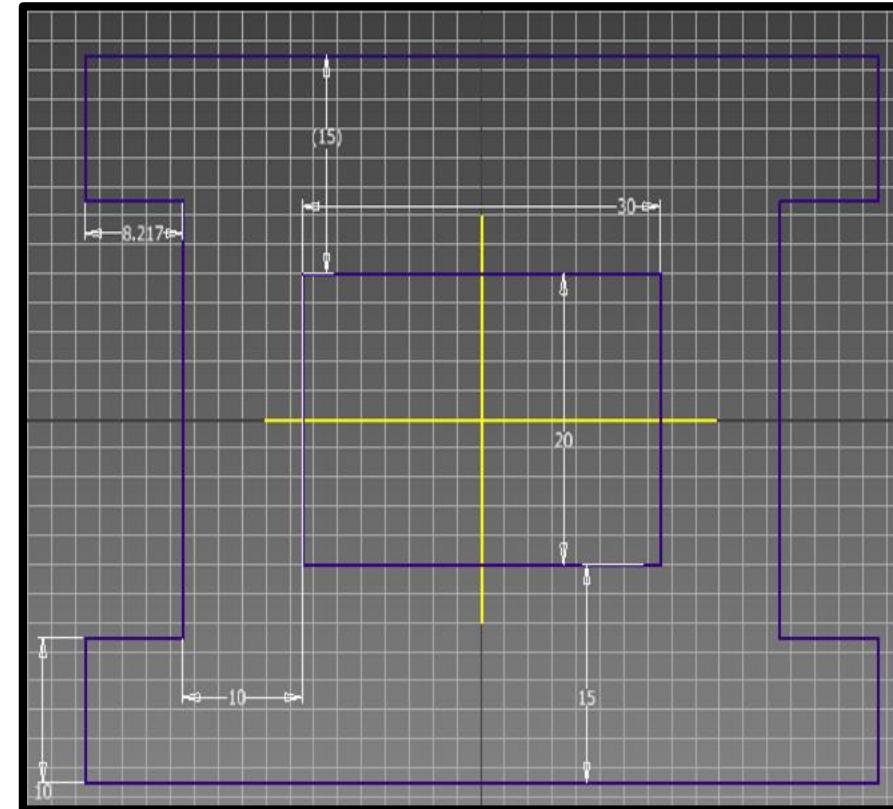
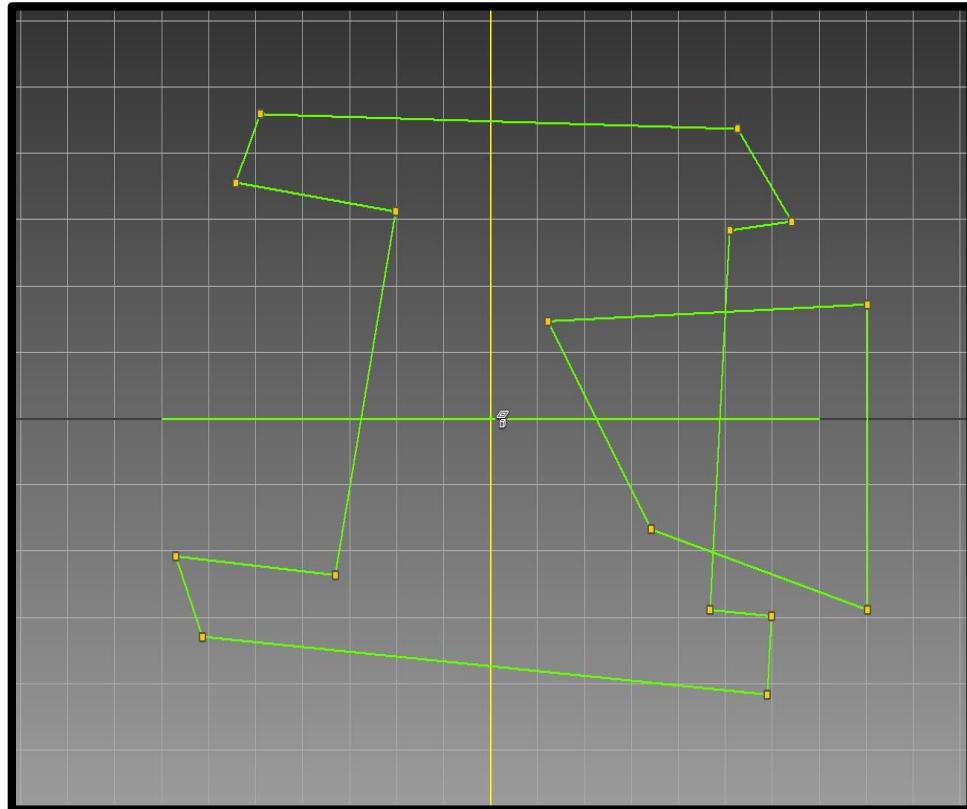
# PRACTICE EXAMPLE 4 (OPTIONAL)

- If you finish early, or want some extra practice.
- Open the **Exercise 4 Guide** and edit the **Exercise 4** file, found in the lesson materials.



# PRACTICE EXAMPLE 5 (OPTIONAL)

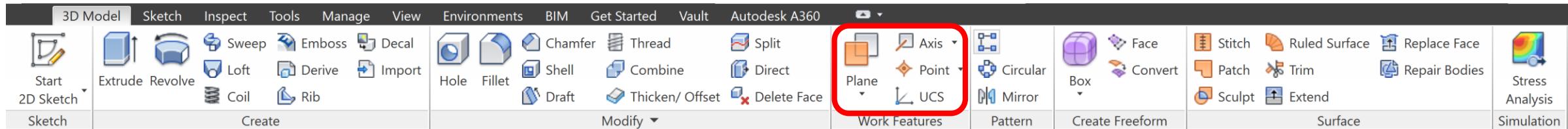
- If you finish early, or want some extra practice.
- Open the **Exercise 5 Guide** and edit **Exercise 5** file, found in the lesson materials.



# 3D MODELLING

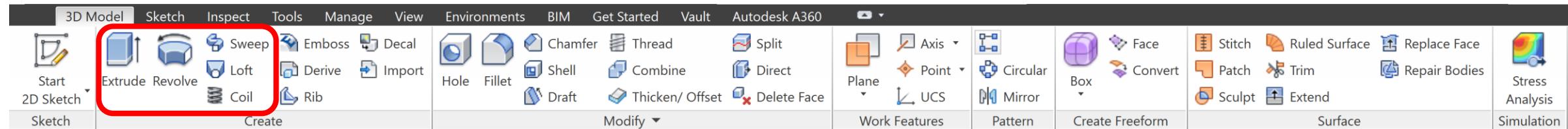
- Using the ***profile*** the 3D modelling command can be used to produce a 3D design.
- The general process for modelling a 3D component is:
  - Create custom work features to establish reference elements, if needed
  - Model with primary and secondary 3D base features.
  - Apply modification features to component, if needed.
  - Apply patterning tools to component, if needed.
  - Apply iProperties and Material Selection.
  - Completed part design.

# CUSTOM WORK FEATURES



- Beside the **origin work features**, the 3D environment is rather bare of reference elements and certainly does not have any away from the origin point.
- **Custom work features** can be generated to solve this issue, these are work features (reference elements) that are designated by the user.
  - Custom work features are various and some are more appropriate to use than others depending on what reference elements exist in your current design, or for what application it is being used for.

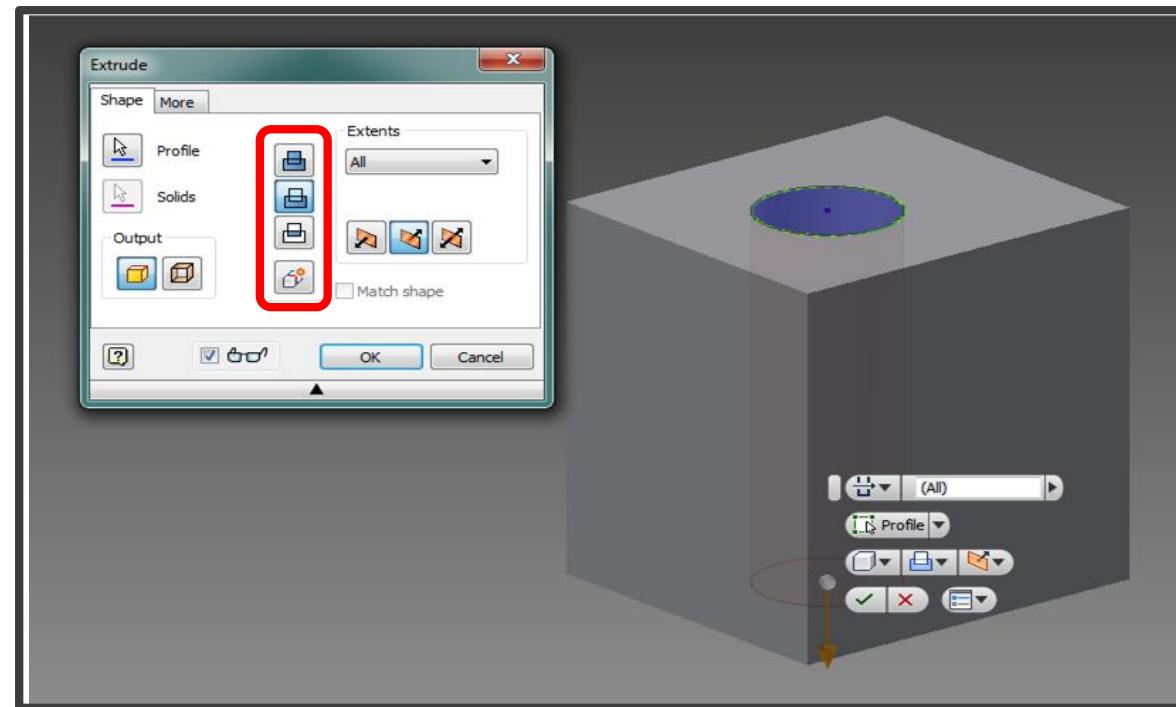
# SOLID BASE FEATURES



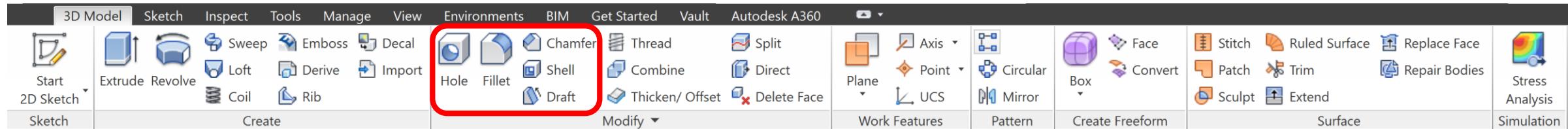
- The solid base feature are the main 3D modelling tools, they create 3D solids based off a 2D profile.
- The basic types are: extrude, revolve, sweep, loft and coil.
  - Each type of 3D base feature is unique in its requirements to execute i.e. the command may require multiple profiles, reference axis, reference path line, etc.
- *The first solid base feature generated can be referred to as 'the primary', all subsequent solid base features are therefore secondary and should build off the primary base solid.*

# SECONDARY BASE FEATURES

- Secondary solid base features are notably different because a new selection menu will become available to denote the type of solid feature created: join, cut or intersect.
- Primary solids do not have this flexibility, they can only be assigned the ***new solid*** type.

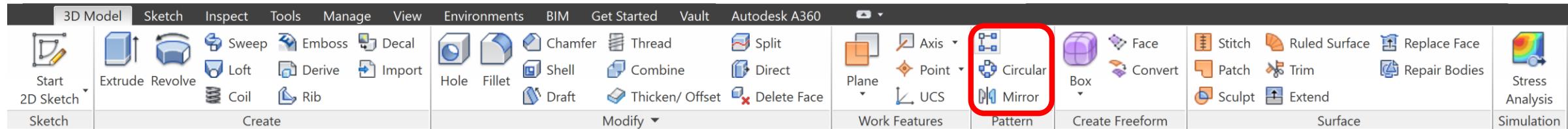


# MODIFICATION FEATURES



- Modification features help to shape solids that cannot be done or harder to model using the 3D modelling features (base feature commands).
- The most prominent types of modifications being: fillet, chamfer, hole and shell.
- *Most notably, the **hole** modification is designed for creating hollow sections that could act as the placement point for a screw or bolt connector, as opposed to simply modelling a **cut** type base feature which is less intricate with its cross-sectional design.*

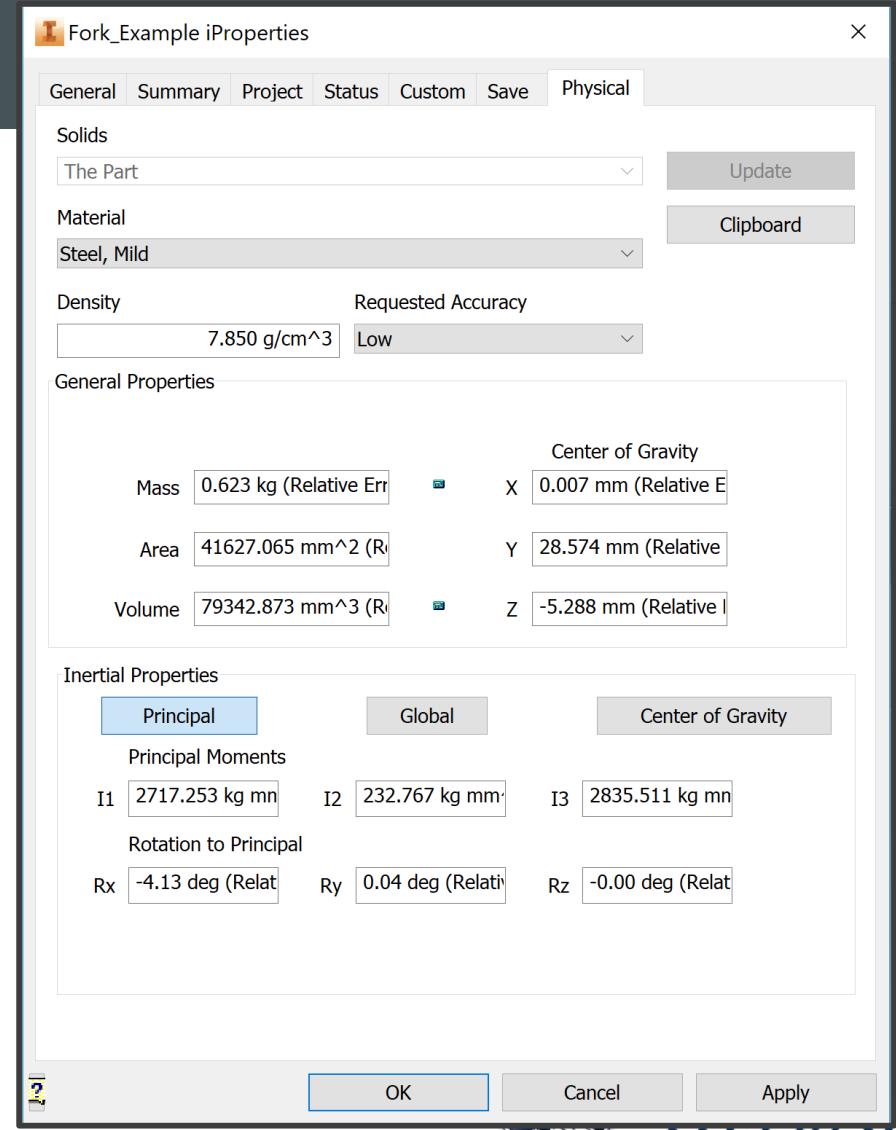
# PATTERNING



- In more complex designs, repeating features can be a tedious process especially when there are many of them, placing a line of bolt holes in a metal plate for example.
- Using the patterning modification commands, features (or groups of features) can repeat using a single operation to make this process more efficient.
- The different types of pattern methods are: mirror, rectangular pattern and circular pattern, to correspond to various design styles.

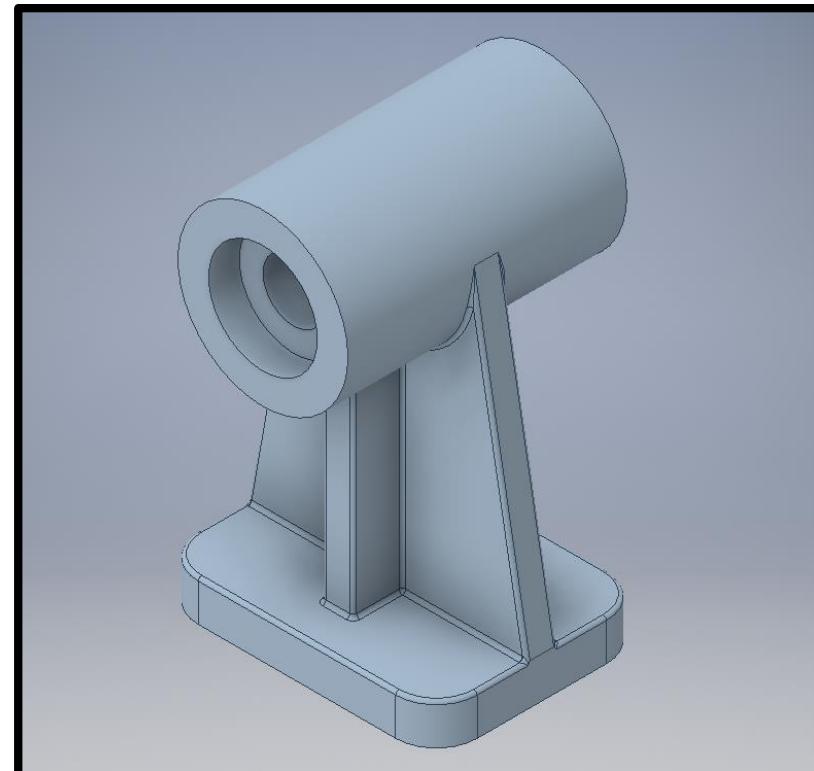
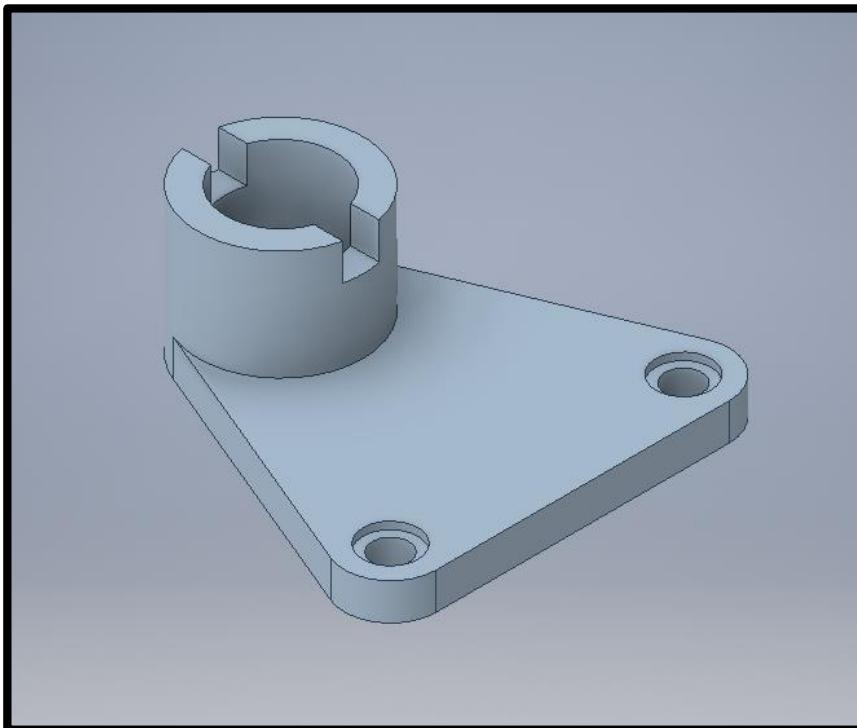
# MATERIAL SELECTION

- Found in the ***iProperties*** menu, selecting a material for the component provides the part with further information.
  - The component will change colour or texture to correspond with the material selection (these can be changed through the colour palette located in the top-ribbon, without changing the material).
  - Physical properties for the part will be automatically calculated based on the selected 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.



# DEMONSTRATION: PARTS MODELLING

- Create the component/s based on the engineering drawing provided.
- **Demonstration 1** (Thursday) and/or **Demonstration 2** (Friday) can be found in the lesson materials.



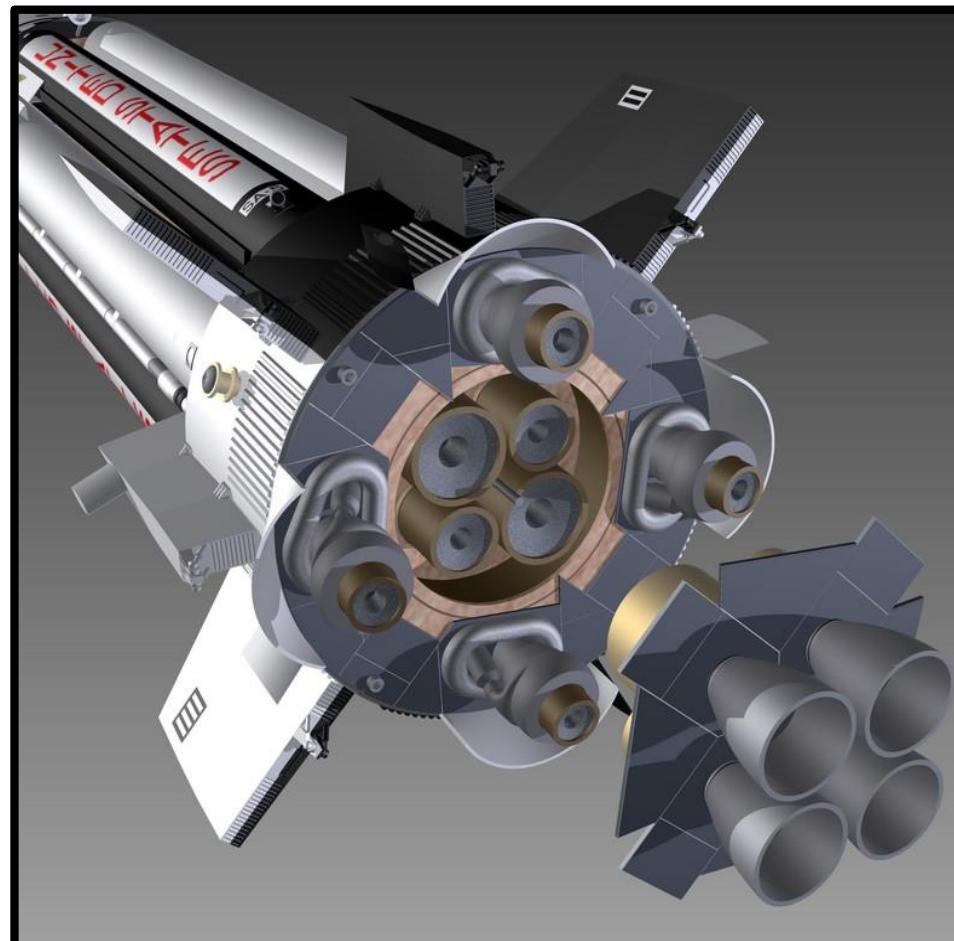


# ASSEMBLY MODELLING

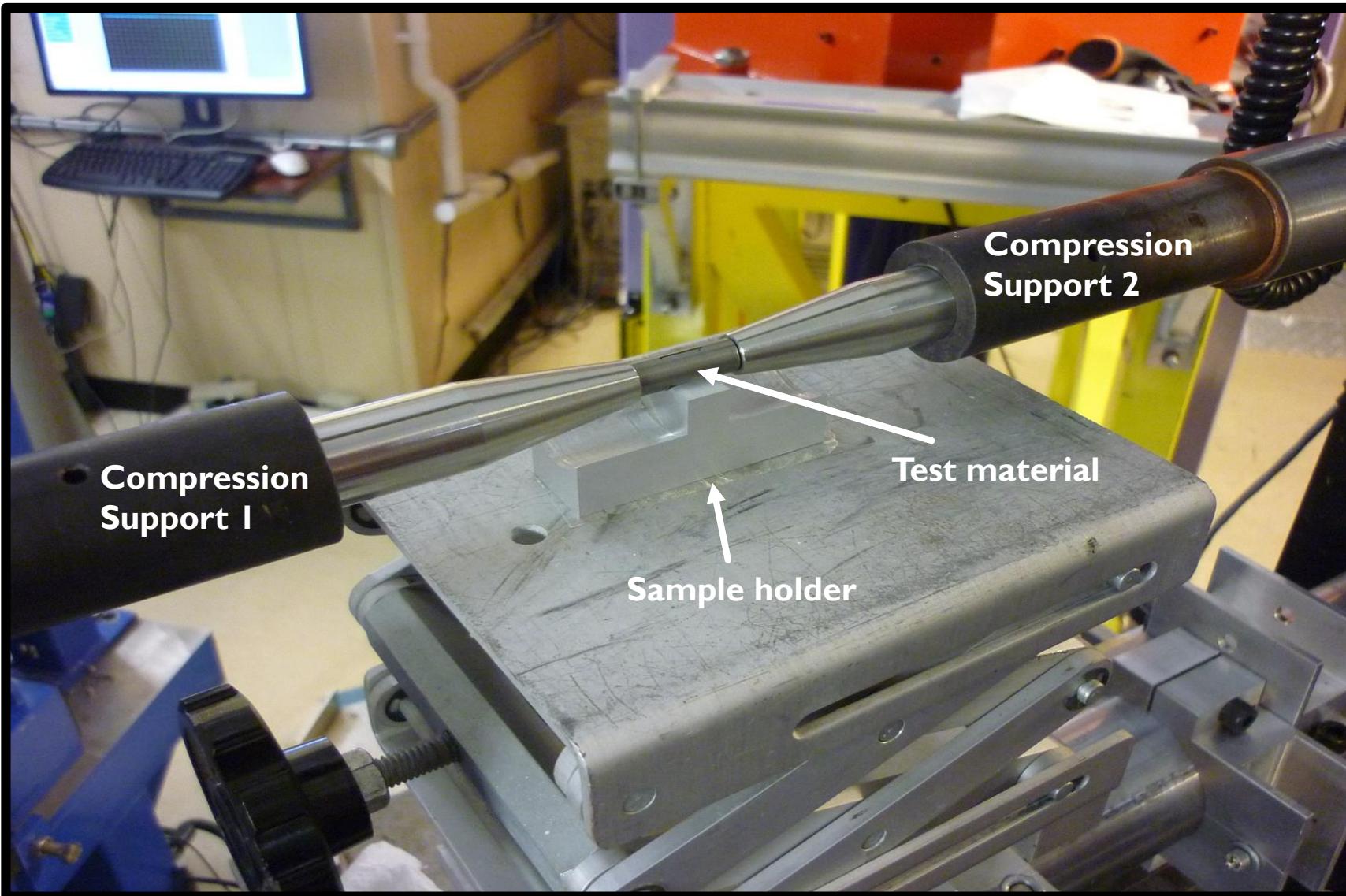
***LESSON 2***

# THE ASSEMBLY MODULE

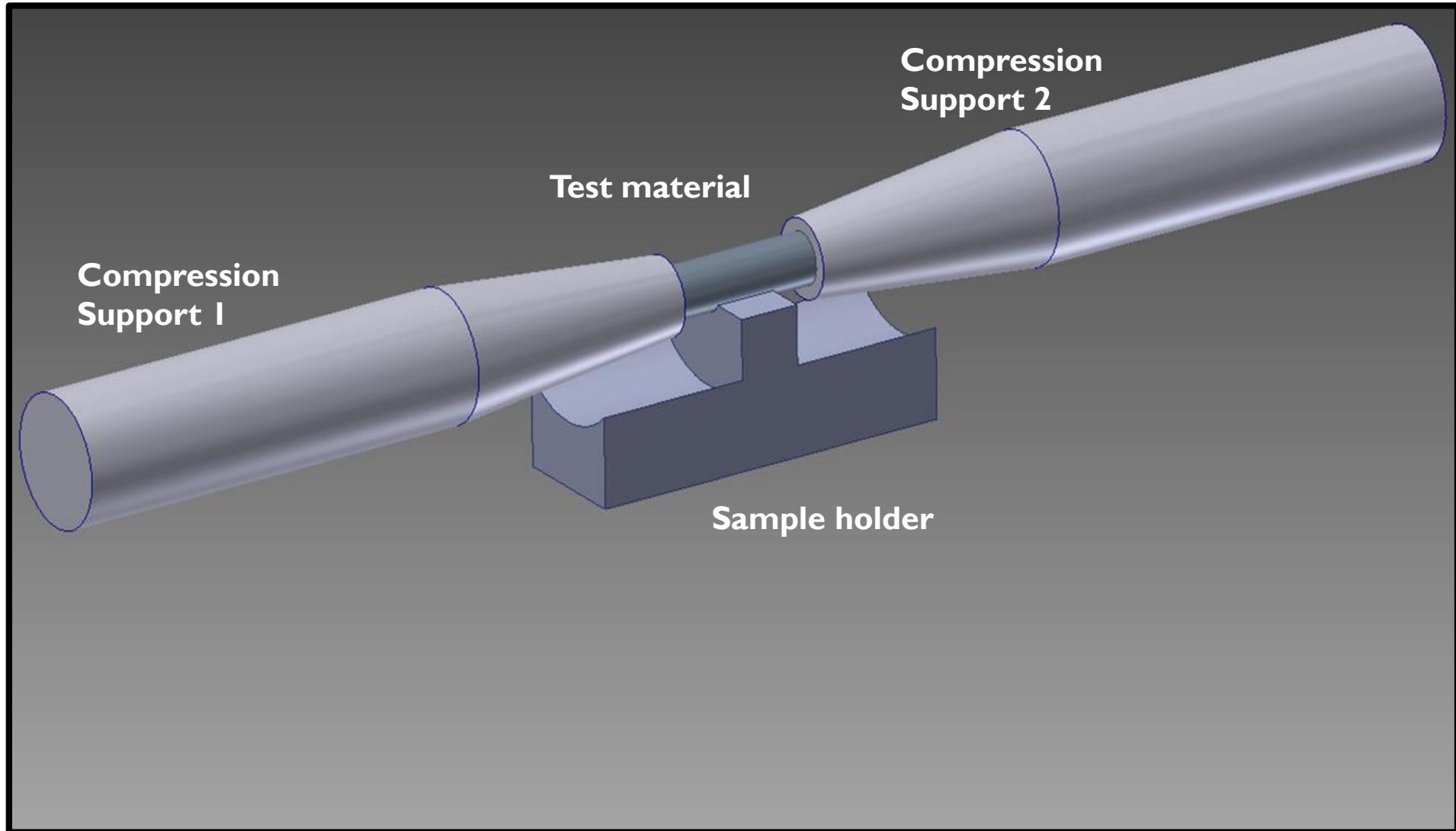
- A method using in CAD software to handle multiple files that represent components within a product design.
- Individual part files are **imported** into the work space and **assembled** together to create the overall product.
- The assemblies module enables users to preview and test the completed product before fabrication.
- Can be used to design apparatus or set ups, i.e. in the example shown.



# EXAMPLE: SAMPLE HOLDER



# MODEL: SAMPLE HOLDER

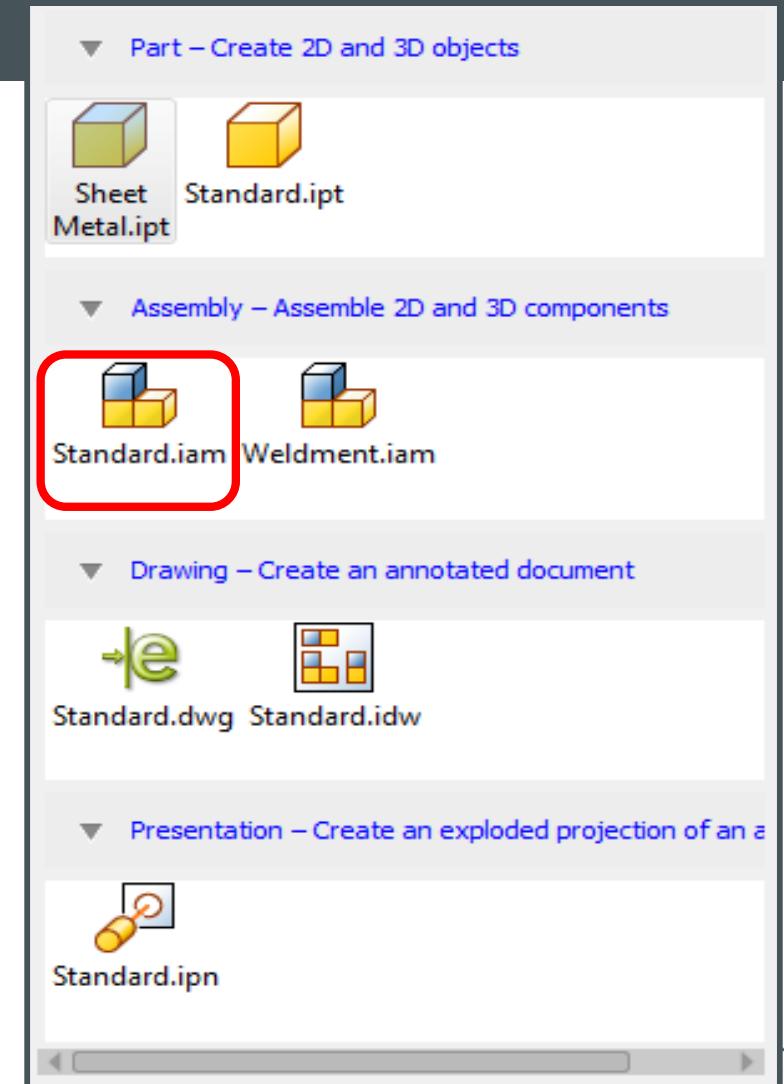


# THE ASSEMBLY MODULE

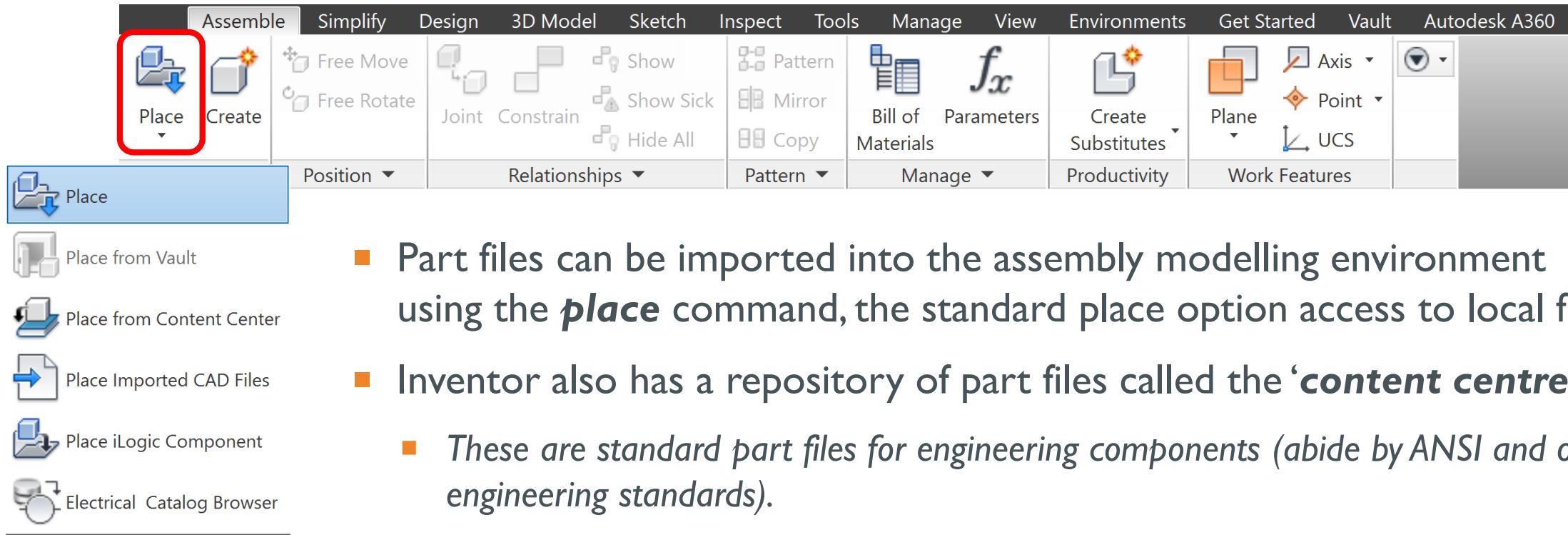
- In the building/designing of projects in the assemblies environment, the general process for assembly modelling is:
  - Import CAD part files
  - Apply relationships to join/constrain components
  - Use measurement tools to determine offset distances and possible interference between components in design.
- We will present a demonstration on assembly modelling, other example are included for your own learning/practice.

# CREATING AN ASSEMBLY FILE

- An assembly file is associated with the **.iam** extension.
- The **standard** .iam assembly files consist of importing part files that can be linked together with **parametric relationships**.
- **Weldment** .iam assembly files assumes that imported part files are **permanently** joined together (using a welded joint).

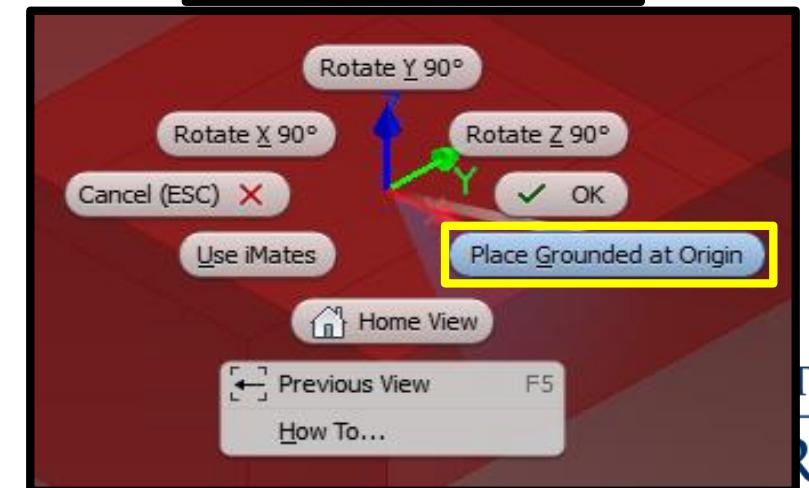
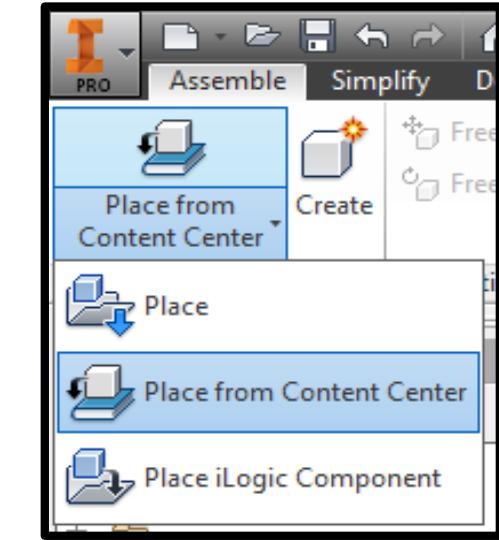


# IMPORTING A PART FILE



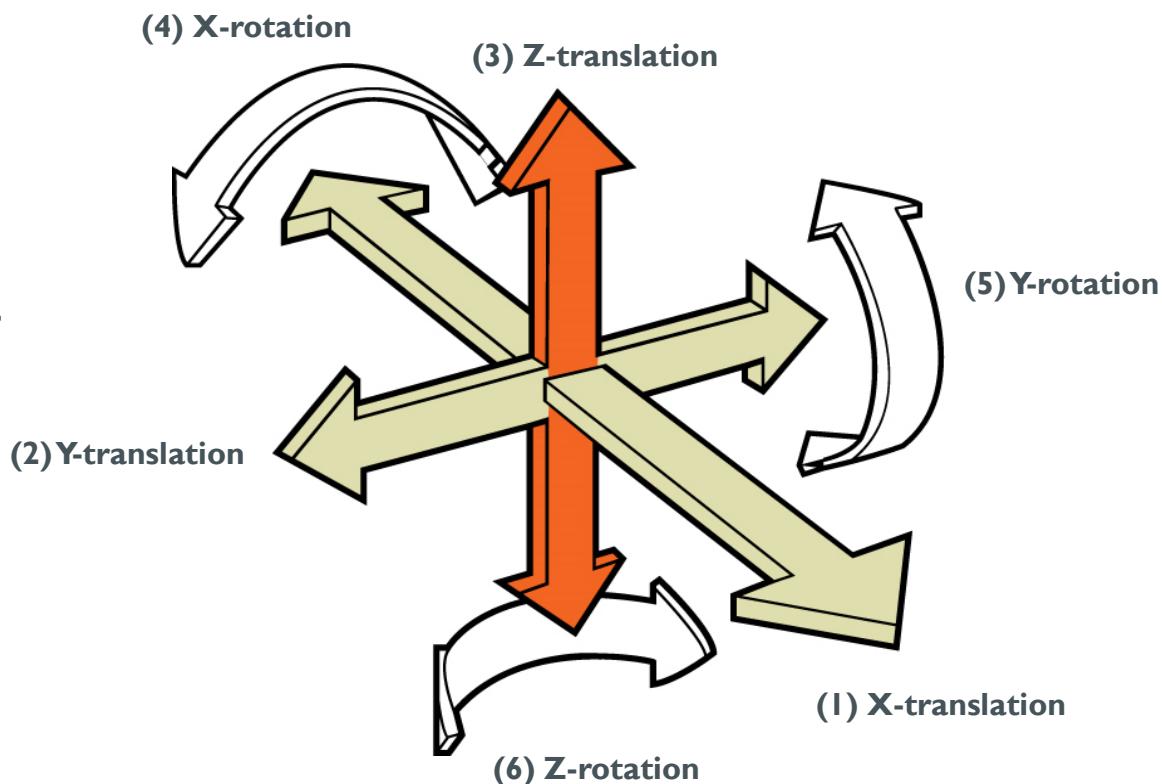
# PLACING PART FILES

- A valid assembly requires the placement of parts (or components). A part that had been placed in the assembly environment is called an ***instance***.
  - It is possible to have multiple instances of the same part in an assembly file.
- The ***sources*** of part files include local files, the inventor content centre and iLogic components (not covered).
- It is good practice to ***ground*** the first part you have placed at the origin. A grounded component will remain fixed to the origin.

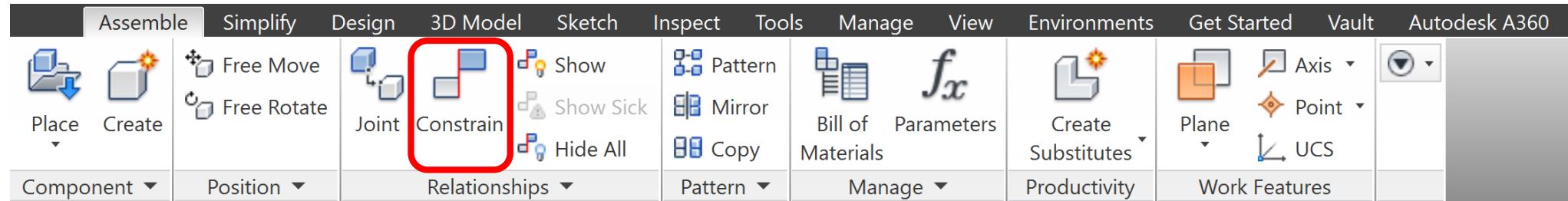


# COMPONENT RELATIONSHIPS

- A single component that has been imported to the assemblies module (unless it is a grounded component) has six degrees of freedom.
- To connect these components we will impose relationships on them to reduce their movement relative to each other, (i.e. reducing their degrees of freedom).
- The two type of relationships that we can apply are: constraints and/or joints.



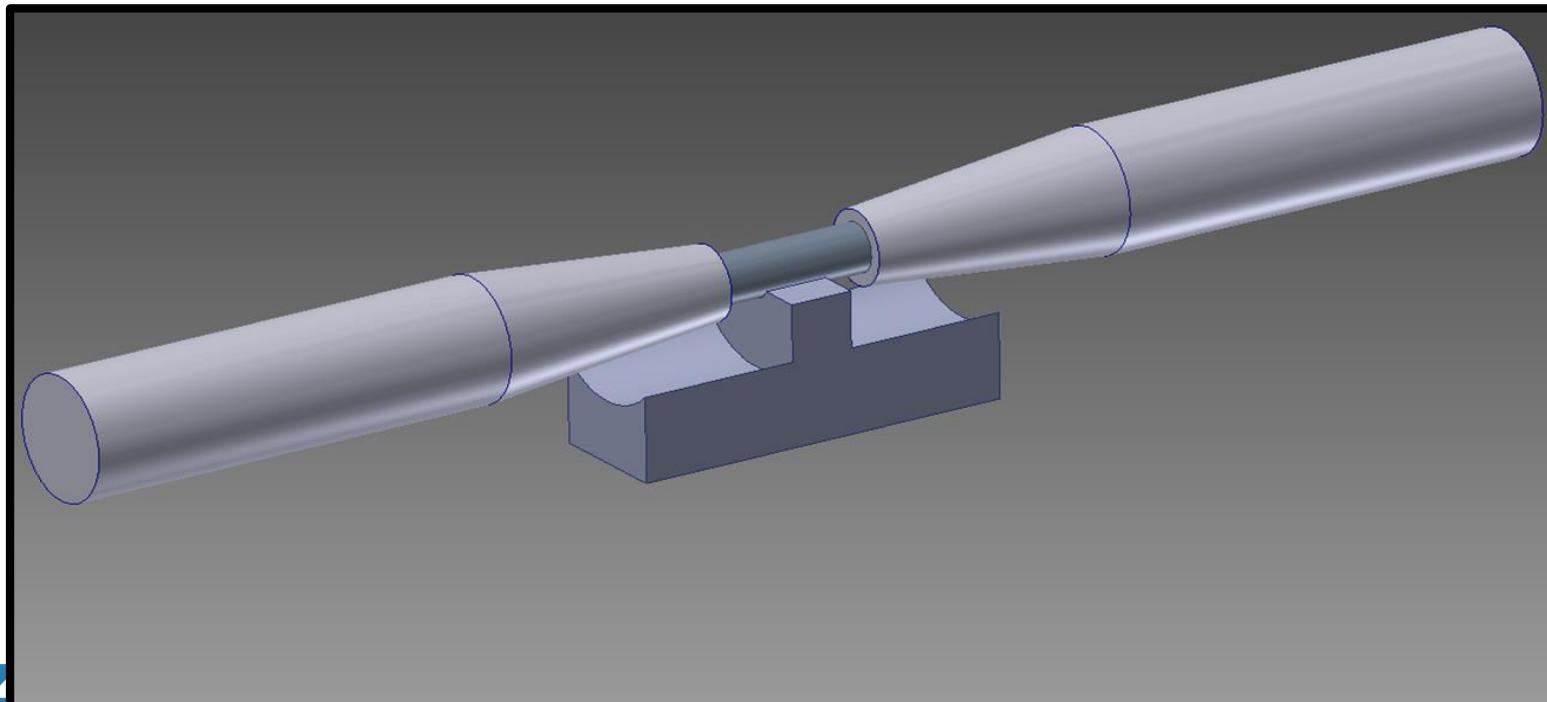
# CONSTRAINT RELATIONSHIPS



- The **constrain** command applies a single relationship between two components.
  - *The constrain method is preferable to applying joint relationships when the assembly of the design is ambiguous or often times does not adhere to a specific type of standard engineering mechanical joint.*

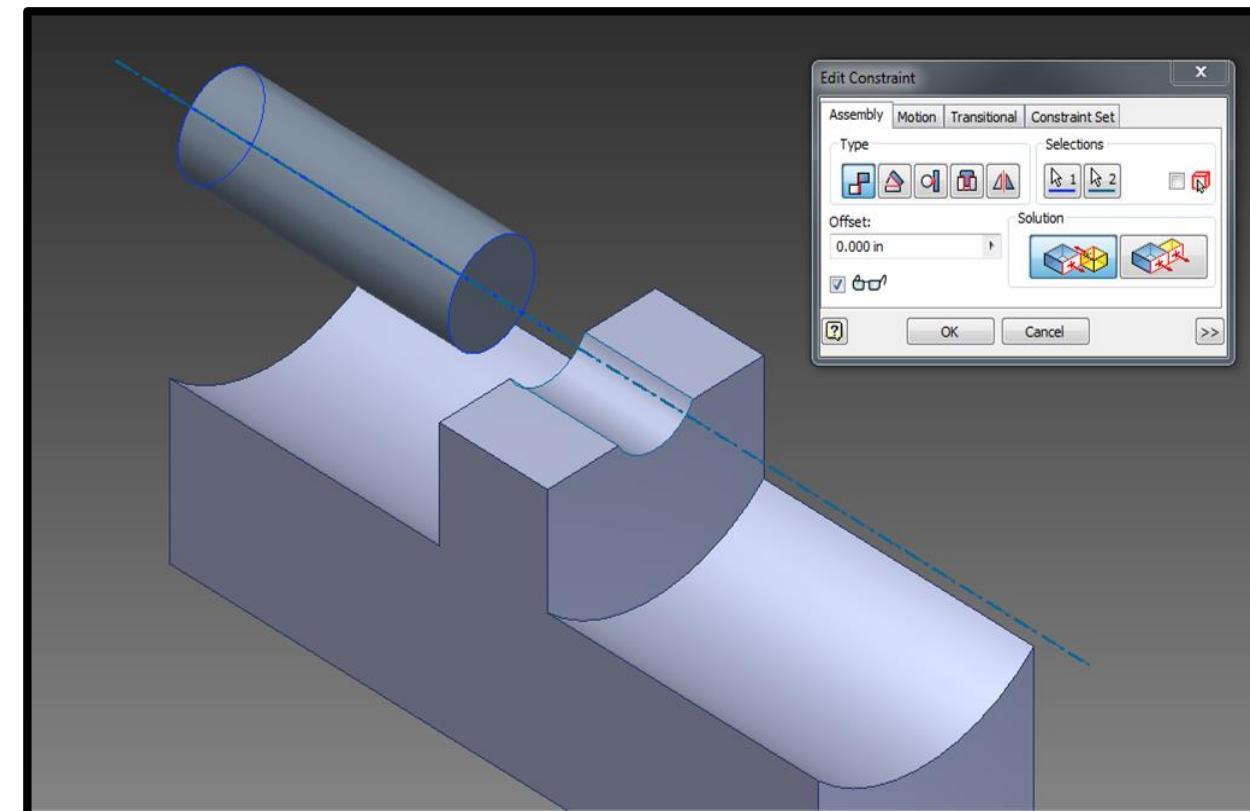
# PRACTICE EXAMPLE 6: SAMPLE HOLDER

- Try to assemble the experiment example using the constraint method.
  - Would it be possible to assemble this design using joints?
- All part files can be found in “Sample\_Holder” project folder of the lesson materials.

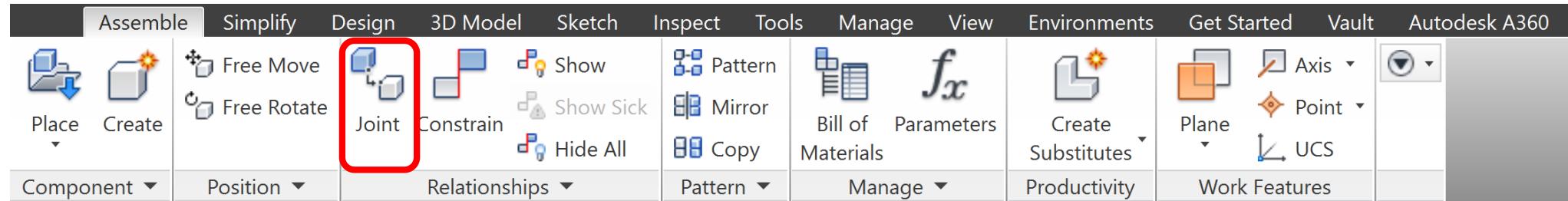


# PRACTICE EXAMPLE 6 REVIEW

- It is possible to constrain **axis** by selecting **curved** surfaces, because *these are surfaces that will not have a distinct surface normal.*
- A few tips that can be helpful:
  - Think about the design first, and determine which part file you will import first as your grounded component.
  - Your grounded component should be the **centre piece** of your design, i.e. you will assemble other components around your grounded component.
  - Maybe import one part file at a time, and fully constrain them when possible to remove clutter from your work space.

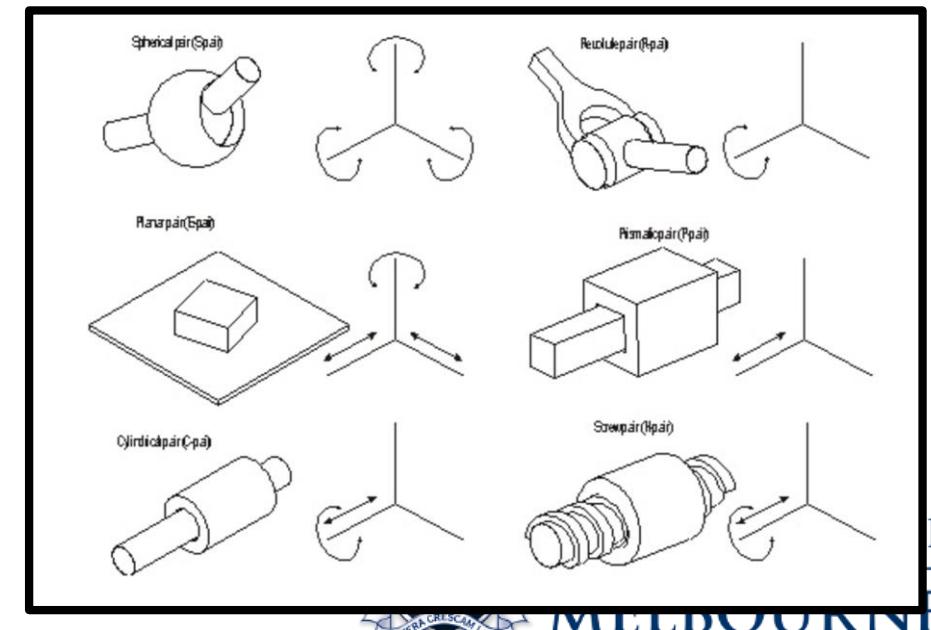


# JOINT RELATIONSHIPS



- **Joint** relationships are a set of constraint relationships that result in specific modes of movement between two components.
  - There are six types of engineering mechanical joints which are also reflected in Inventor
- The joint method is more applicable to designs where there is a clear understanding or expectation in how the final assembly will move.
- Inventor has an inbuild **automatic** joint option that will assume a certain type of joint based on the reference elements selected.

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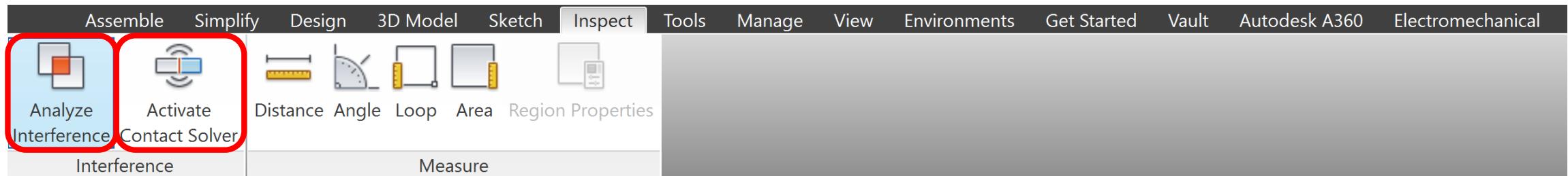


# MEASUREMENT TOOLS



- To ensure that designs are assembled with accuracy, a number of parameters can be analysed using the ***measurement tools***.
- The measurement tools can be used to analyse the dimensions of components to understand ***how*** certain components joint together.
- They can also be used to measure offset distances for specific tolerance or fit types.
  - *Inventor also allows components to overlap which would not occur with physical components, thus it is also possible to check for interference.*

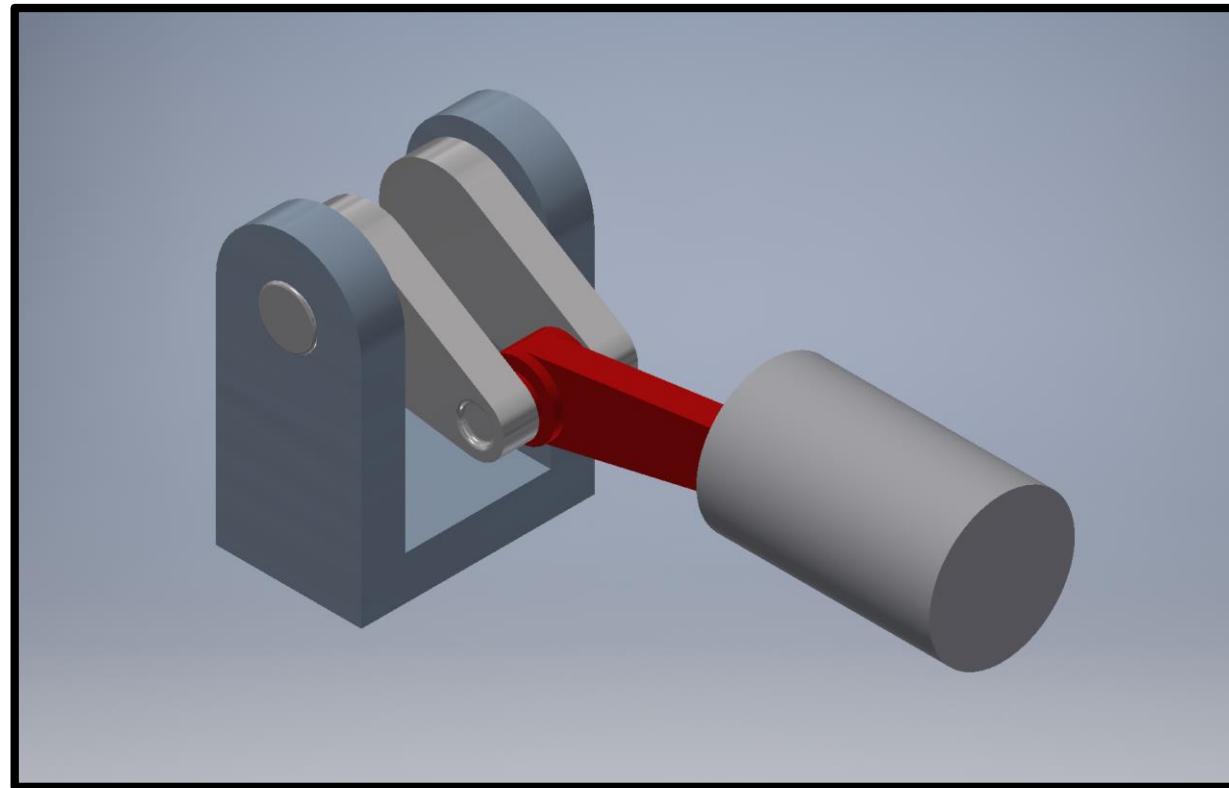
# INTERFERENCE & CONTACT SETS



- The **analyse interference** command will determine if there is any overlap between components in the current configuration. However, in dynamic systems component will move and thus interference detection will need to be determined for each point in its motion (path of movement).
- The creation of a **contact set** will impose physical constraints on a set of components such that they cannot overlap with each other once the **contact solver** is turned on.
  - Create a contact set by group selecting the components you want in the set from the model browser, right-click to open the menu and select the contact set option.

# DEMONSTRATION: ASSEMBLY MODELLING

- Assemble the mechanical system using joints and/or constraint relationships, and then check for any interference in the components.
- *All part files found in “Piston\_Example” project folder of the lesson materials.*





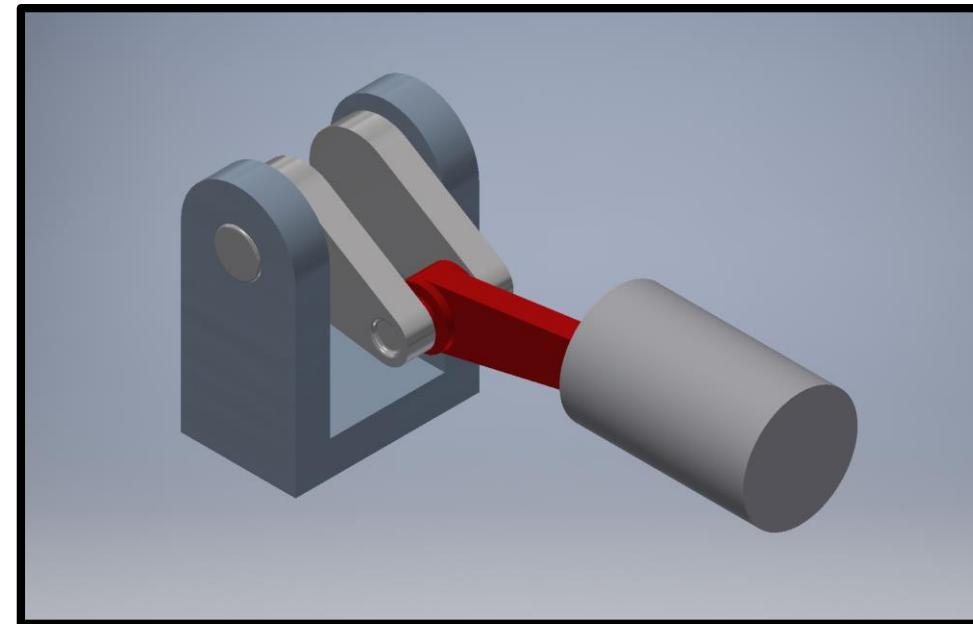
# MOTION SIMULATION & EXPLODED PRESENTATIONS

**LESSON 3**

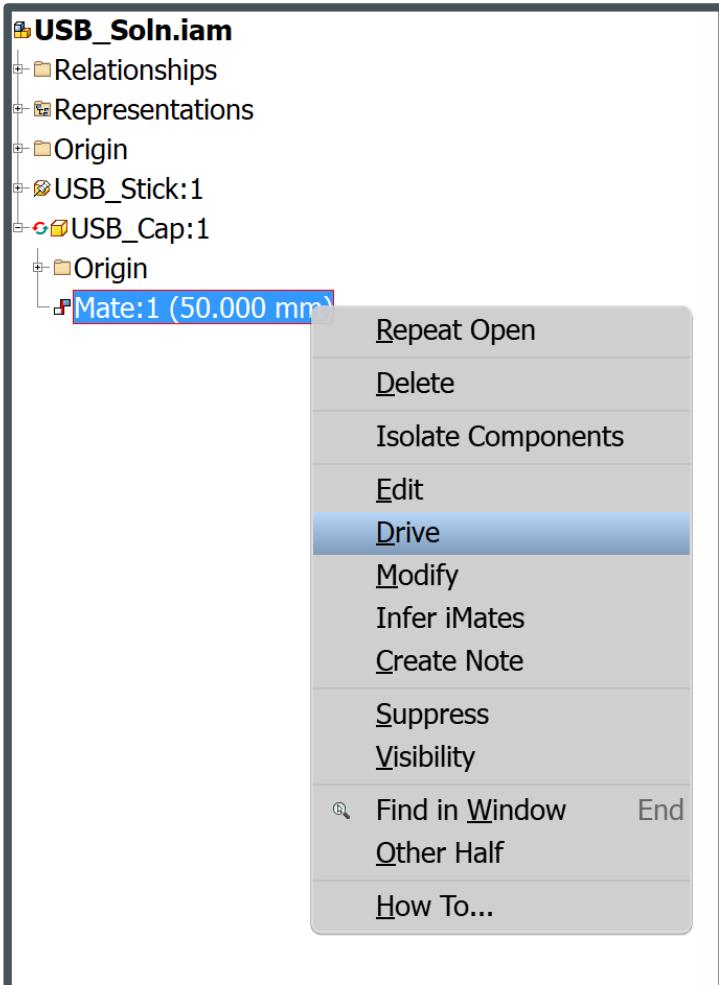


# THE ASSEMBLY MODULE (MOTION SIMULATION)

- This lesson will have two components, first we will return to the *piston\_example* to demonstrate a motion simulation.
- We do this because, in project design we want to understand how well crafted a model/design is.



# MOTION SIMULATION



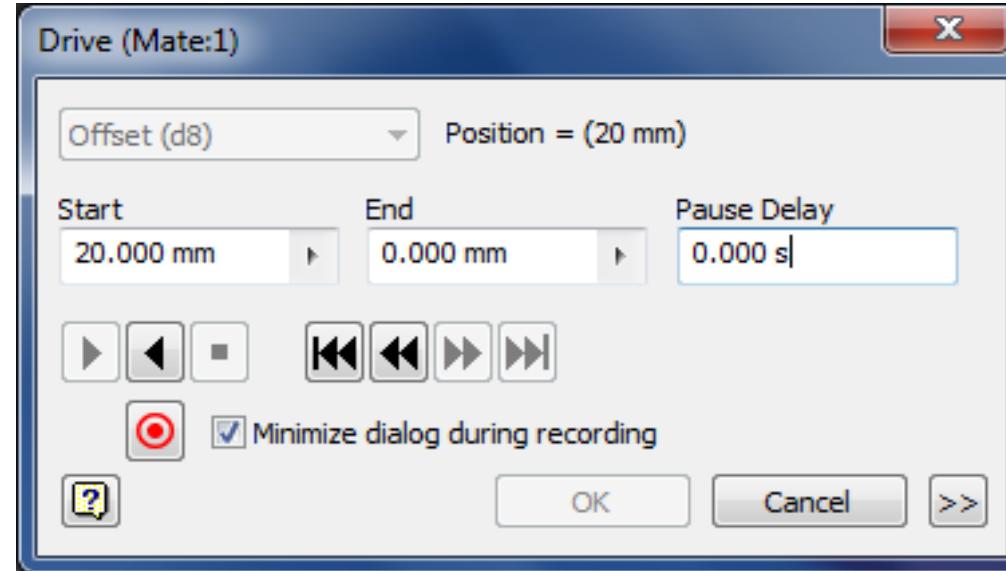
- To simulate functionality, assemblies can be **driven** through their joints or constraint relationships.
- *The driven constraints command can be found by right-clicking the relationship in the model browser and selecting the **drive** option.*



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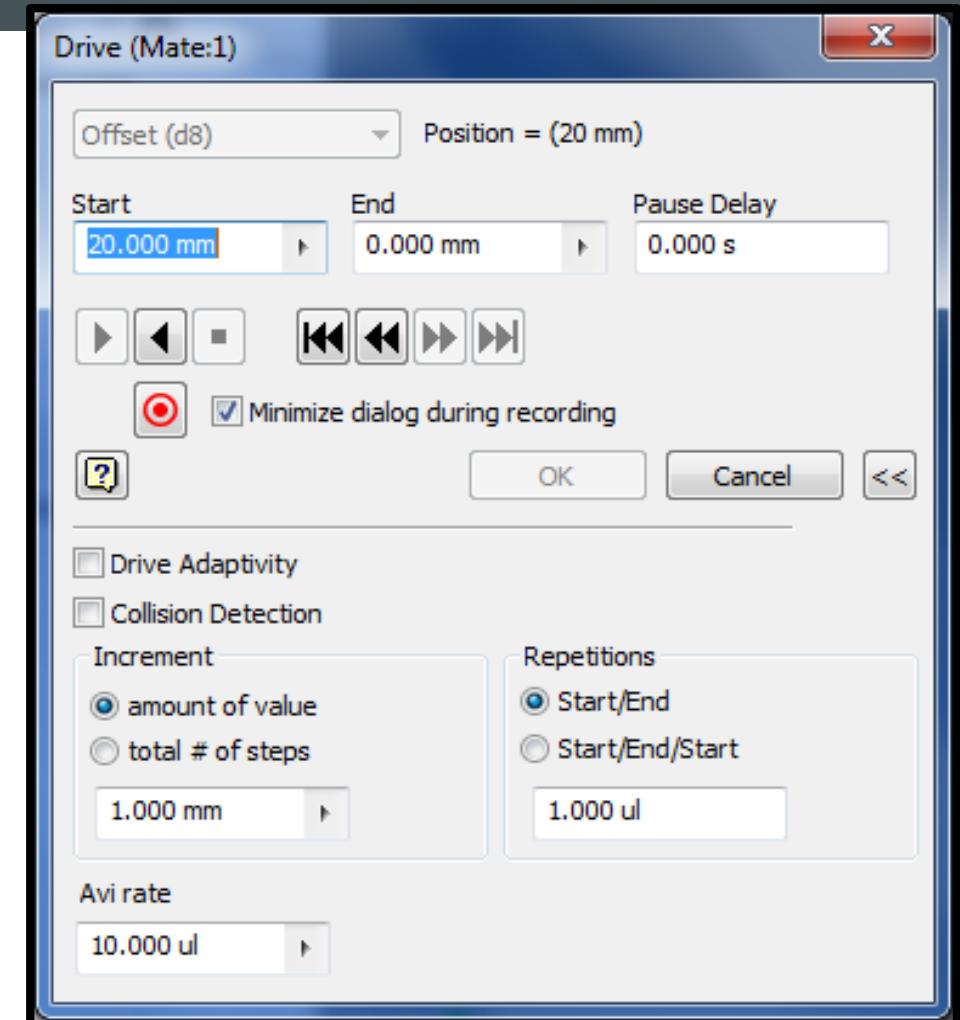
# DRIVEN CONSTRAINTS

- Constraints can be driven in the **forward** or **reverse** direction, moving the components between a **start** and an **end** position.
- A pause delay can be added to slow down the motion if needed.
- The simulation motion can also be recorded to produce videos.



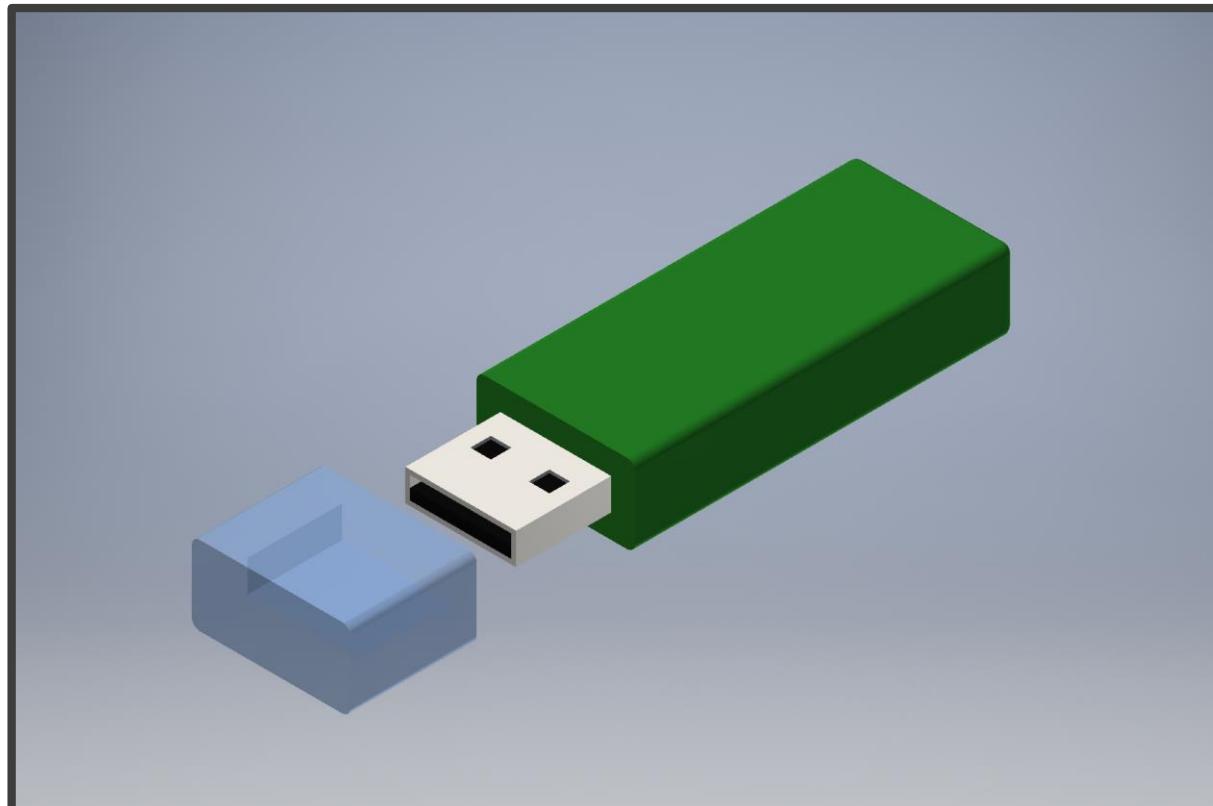
# DRIVEN CONSTRAINTS

- In the advanced settings you have more control over the ***time intervals*** and ***increments*** of the driven motion.
- And can repeat the drive, to put the simulation on a repetitive loop.
- ***Drive Adaptivity***: adapts components while maintaining the constrain relationship.
- ***Collision Detection***: drives the assembly until a collision is detected. When an interference is detected, it is displayed and its value shown.



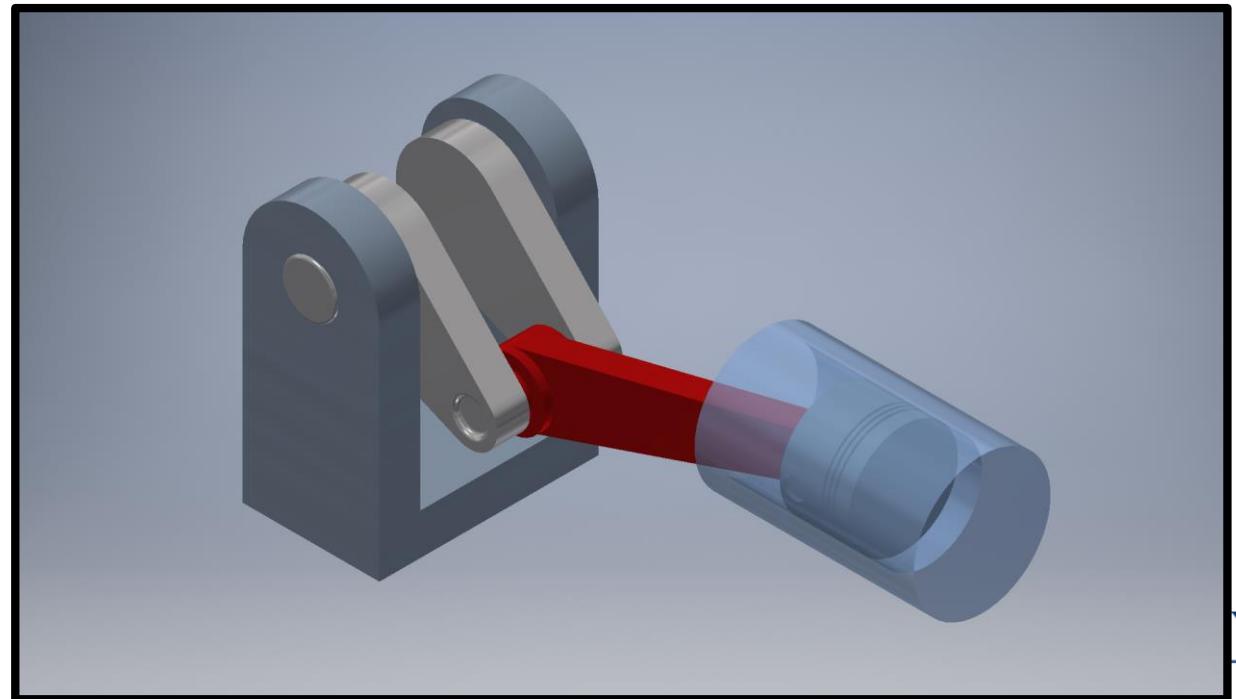
# PRACTICE EXAMPLE 7: USB

- Assemble the example and simulate the action of putting the cap onto the USB.
- *All part files can be found in “USB\_Example” project folder of the lesson materials.*



# DEMONSTRATION: MOTION SIMULATION

- Produce a motion simulation of actuating the piston inside the cylinder.
- Open the **Piston\_Model-Soln.iam** file in the lesson materials.
  - Edit the “flush:1” constraint to an offset value of 185mm, this will remove the interference for the contact set.
  - Change the cylinder material to blue-glazing so that the motion of the piston is visible.

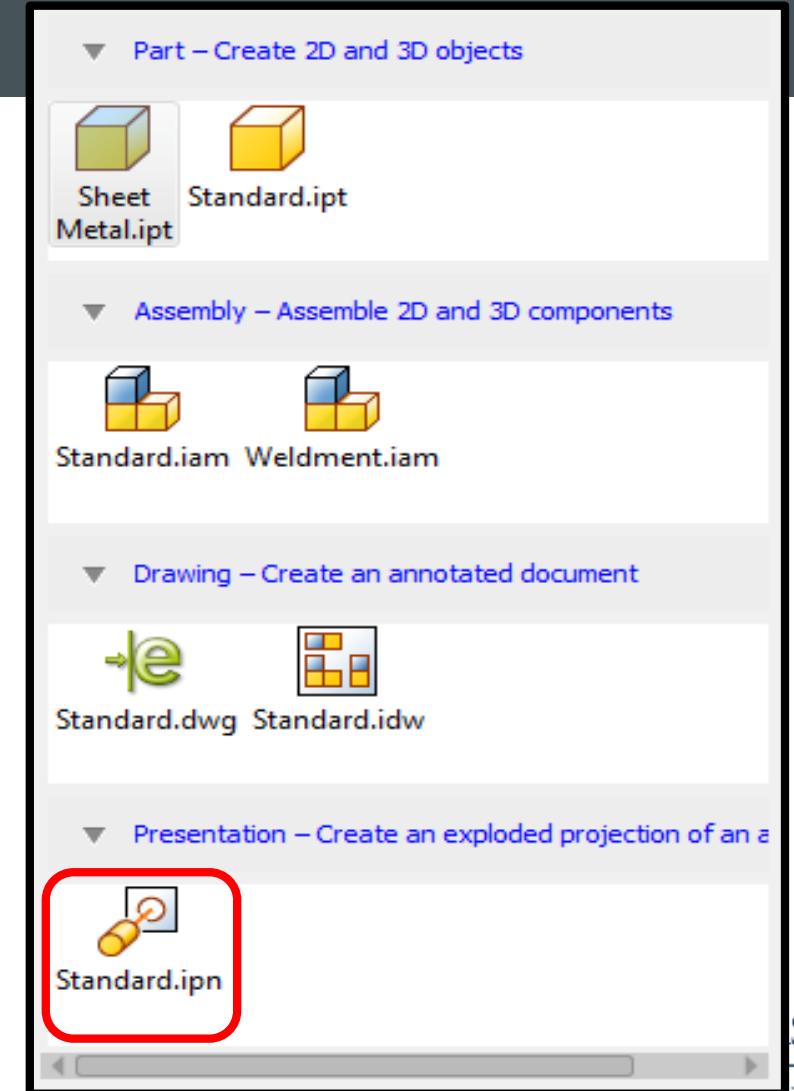


# THE PRESENTATION MODULE

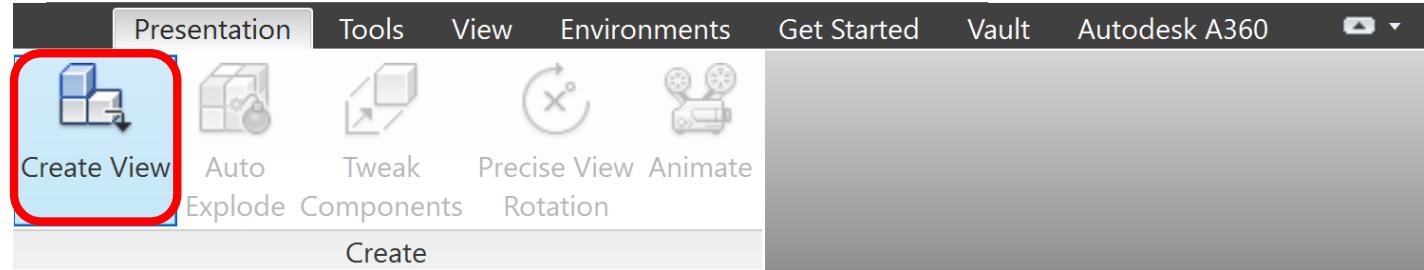
- A presentation of an engineering design make ideas clearer to colleagues or clients, showing: what does the design do, how to make it, etc.
- We will present a demonstration one type of presentation (an exploded presentation).
- The general design process for creating an exploded presentations is:
  - Import CAD assembly file
  - Tweak components to different positions/orientations
  - Animate the process to create the exploded presentation

# CREATING A PRESENTATION FILE

- A presentation file is associated with the **.ipn** extension.
- The presentations module starting with the Inventor 2016 version and newer have been rebuilt to further develop workflow improvements for building exploded presentations and other types of assembly instructional views.

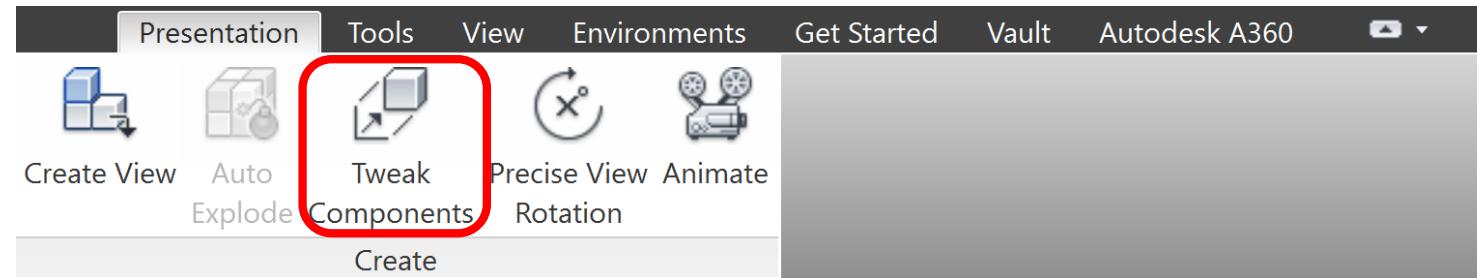


# IMPORT AN ASSEMBLY FILE



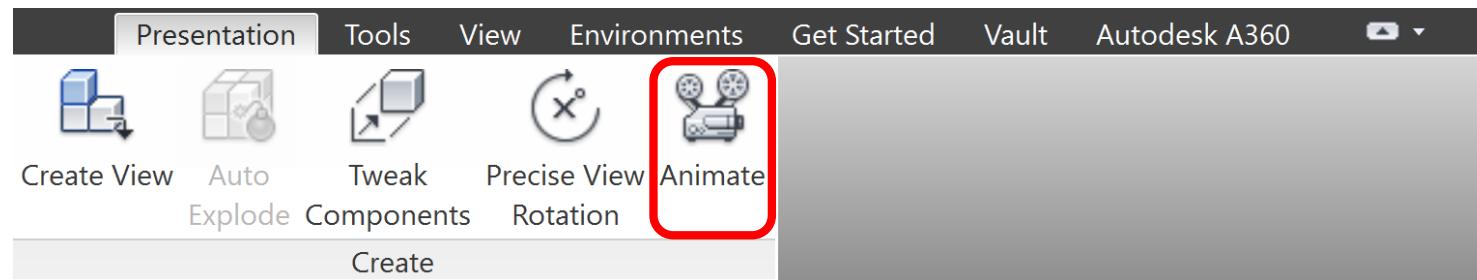
- To import an engineering design into the presentation environment select the ***create view*** command.
- The program can only pull assembly files that happen to be saved in the computer local files.

# MOVING COMPONENTS



- Components can be moved around through the **tweak components** command.
- In selecting a component to move, option will be provided to move that component in a certain direction, rotation, and it is also possible to group select components to be moved together.
- The path lines that appear after a component has been moved is called a trail.
  - *The visibility of trails can be turned off, right-click to bring up the menu option.*

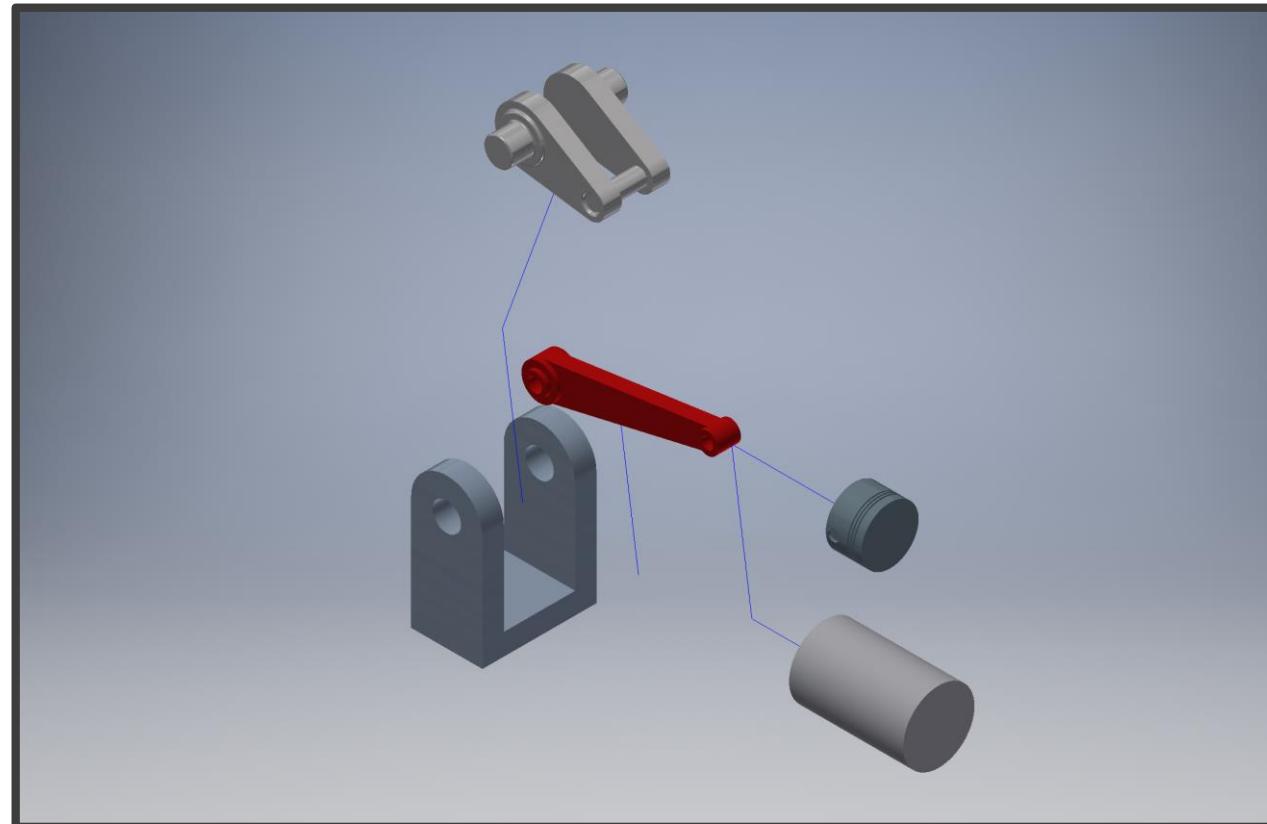
# ANIMATING PRESENTATION



- To create the exploded presentation select the ***animate*** command which will bring up the ***animation*** menu.
- The animation menu will provide the options for a playthrough of the generated exploded presentation.
- Recording the presentation will produce a video recording that can be saved.

# DEMONSTRATION: EXPLODED PRESENTATION

- Create an exploded presentation using the **Piston\_Model-Soln.iam** file in the lesson materials.





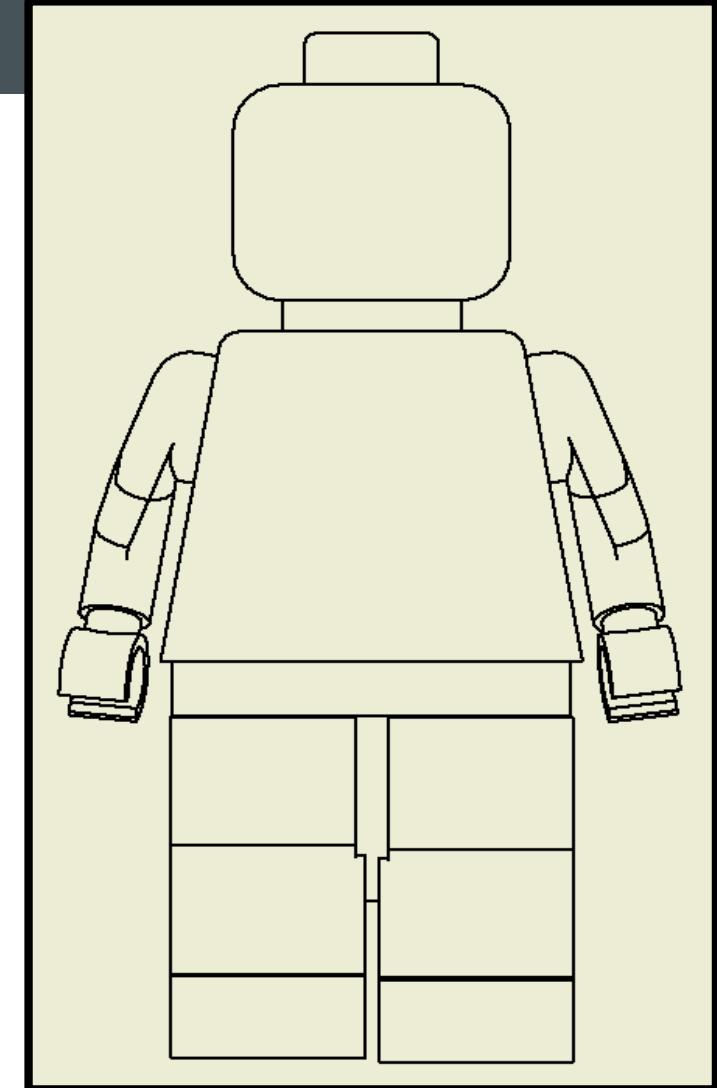
# TECHNICAL DRAWINGS

**LESSON 4**



# THE DRAWINGS MODULE

- A drawing of an engineering design can communicate important information about it, including: manufacturing, purchasing, customer service, etc.
- The drawings module documents components that have been made in the **parts of assemblies** modules as 2D drawings or images.
- We will demonstrate the techniques for producing a technical drawing of a part (component) and an assembly (design).

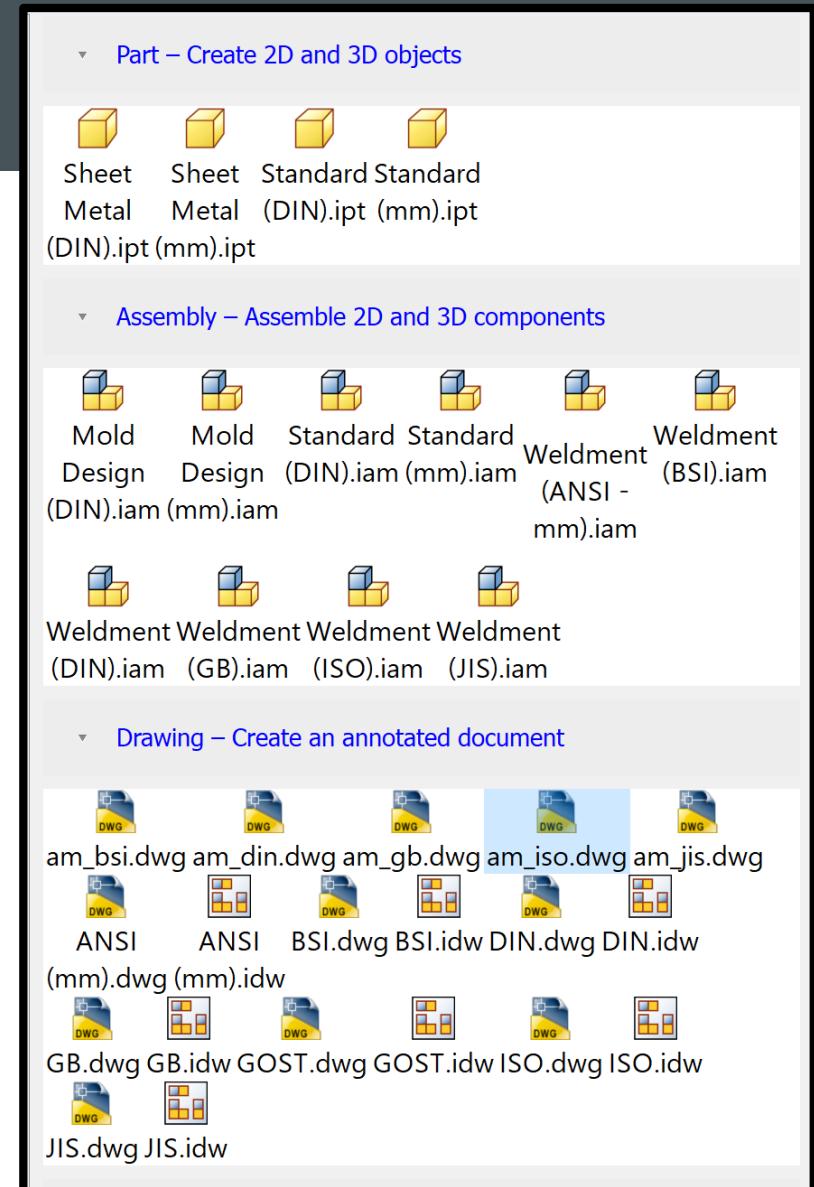


# THE DRAWINGS MODULE

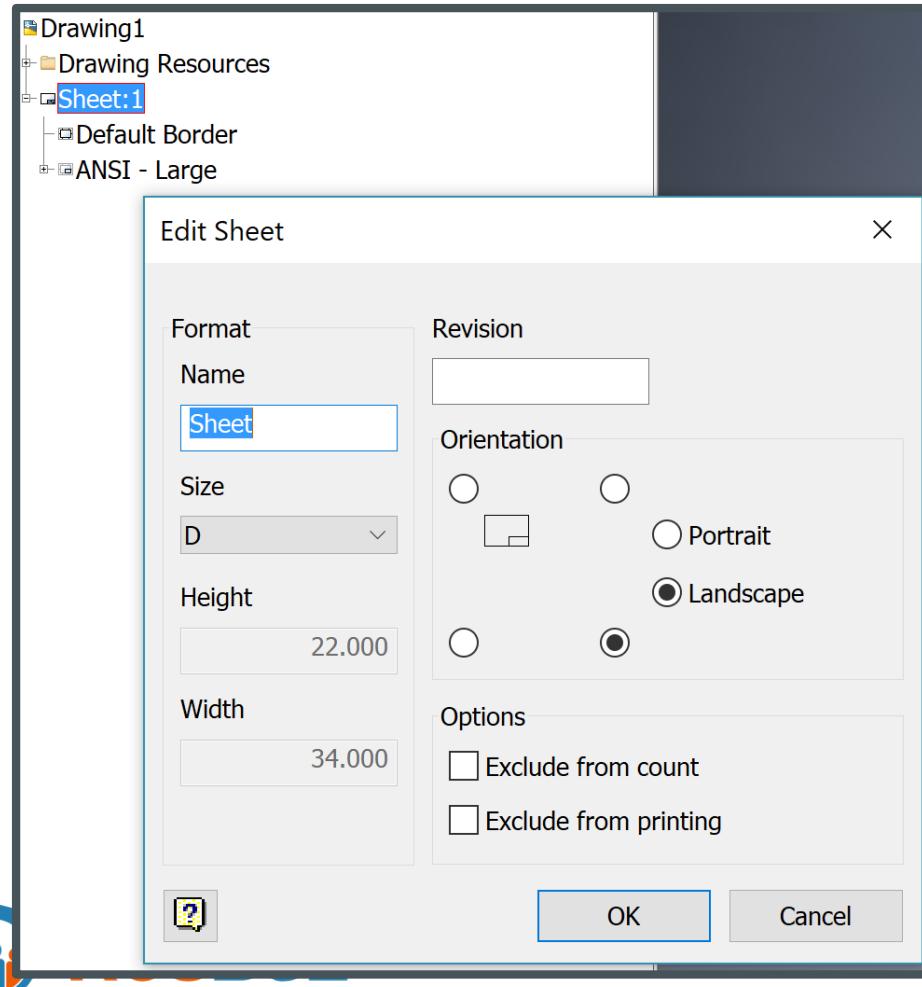
- The general process creating a technical drawing is:
  - Create a new drawing file.
  - Select sheet type.
  - Insert base view.
  - Insert secondary view (using base view as references), including: sectional and detail view.
  - Annotate drawing with dimensions.
  - Apply iProperties (set author and title information).
  - Export file as pdf or image (.jpeg, .png, etc.).

# CREATING A DRAWING FILE

- A drawing file is associated with the **.dwg** or **.idw** extension
- The available options for drawings are set to a number of common engineering standards.
  - ANSI
  - DIN
  - ISO

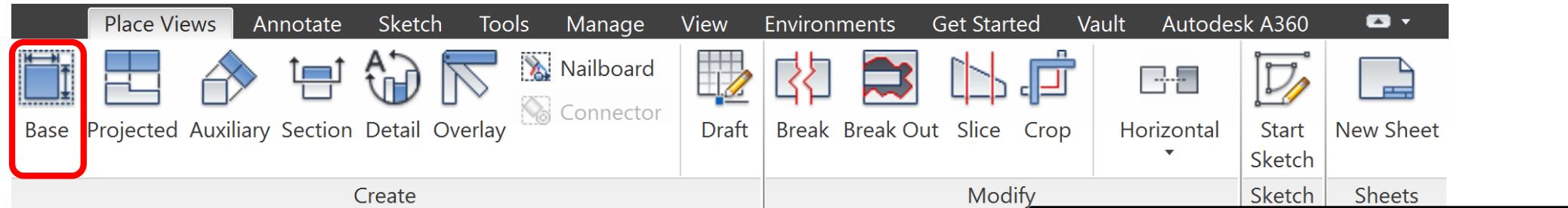


# DRAWING SHEET

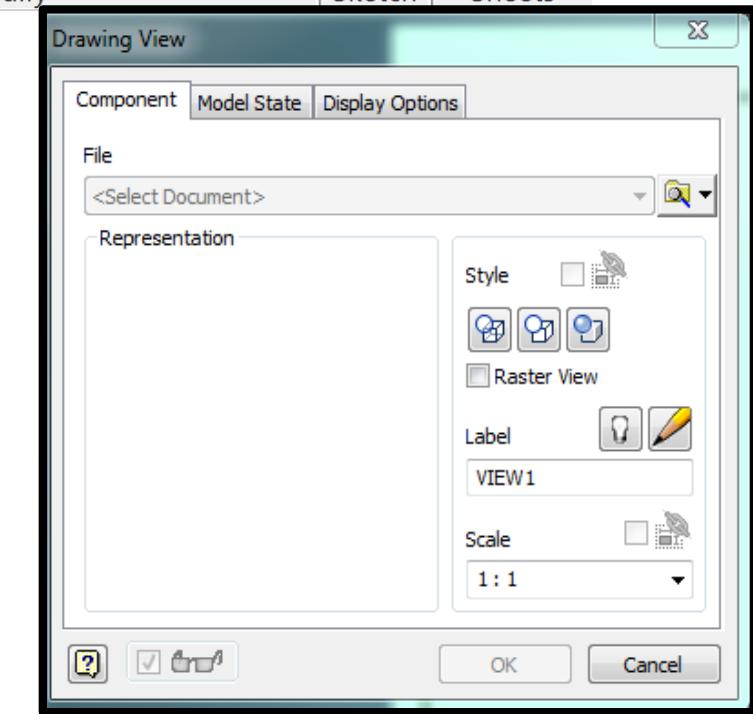


- The drawing sheet is the format of paper that the drawing will appear on, sheets come in a range of different shapes, sizes and styles.
  - The sheet will appear in the graphics window, with its type indicated in the properties field (bottom-right)
  - The drawings module is often default set to using the size D type.
- *To change sheet type, right-click on the sheet item in the model browser and select the **edit sheet** option.*

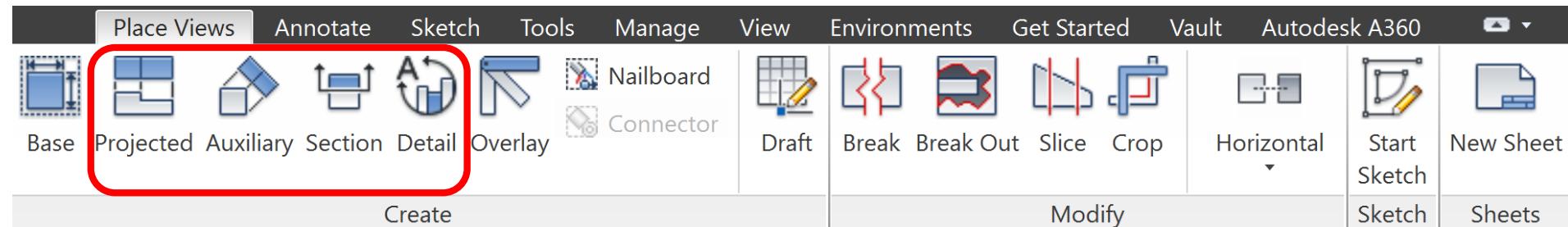
# BASE VIEW



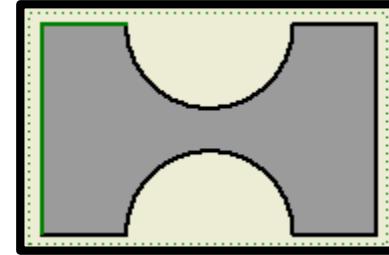
- A **base** view produces the first image/view of the drawing.
- The **orientation** of the view can be adjusted using the navigation cube that appears.
- There are **visual styles** to base view drawings: hidden lines shown or not shown, and shaded
- A **scale** option is available to adjust the size of the view.



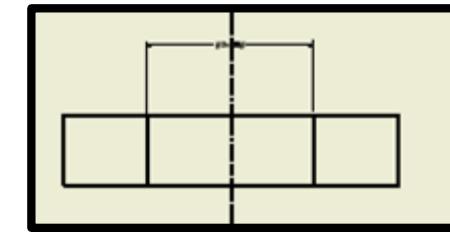
# SECONDARY VIEW



- Secondary views are **dependant** on the base view because they are extended or projected view types that use the base drawing as a reference.
- They types of secondary views include: projected, auxiliary, sectional and detailed.
  - Sectional view will present a cross-sectional view based on where a cut is made.
  - Detail view presents a magnified view of a small section.

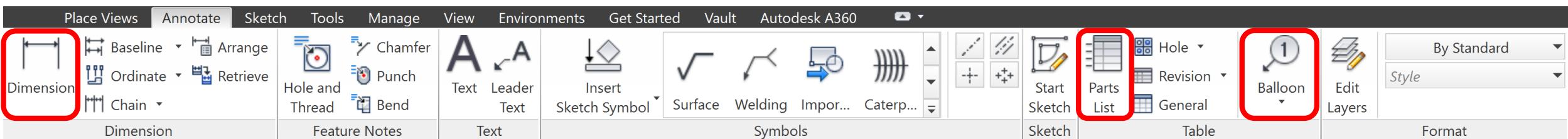


**top view**



**side view**

# ANNOTATIONS



- **Annotations** are text that accompany the engineering drawing in communicating information about the design, most commonly used for giving **dimension** values.
  - The method for placing dimensions is the same as in the parts module when we placed dimensions onto sketches.
  - When producing drawing files for assemblies, the **balloon** and **parts list** commands are useful in identifying different components.

# DRAWING PROPERTIES

- Similar to *iProperties* for a part or assembly file, drawing files need information that identify it as important or useful to the project.
- This information is displayed in the bottom-right corners or any standard engineering document.
- Filling out the required information in the iProperties tab will automatically populate the fields in the drawing.

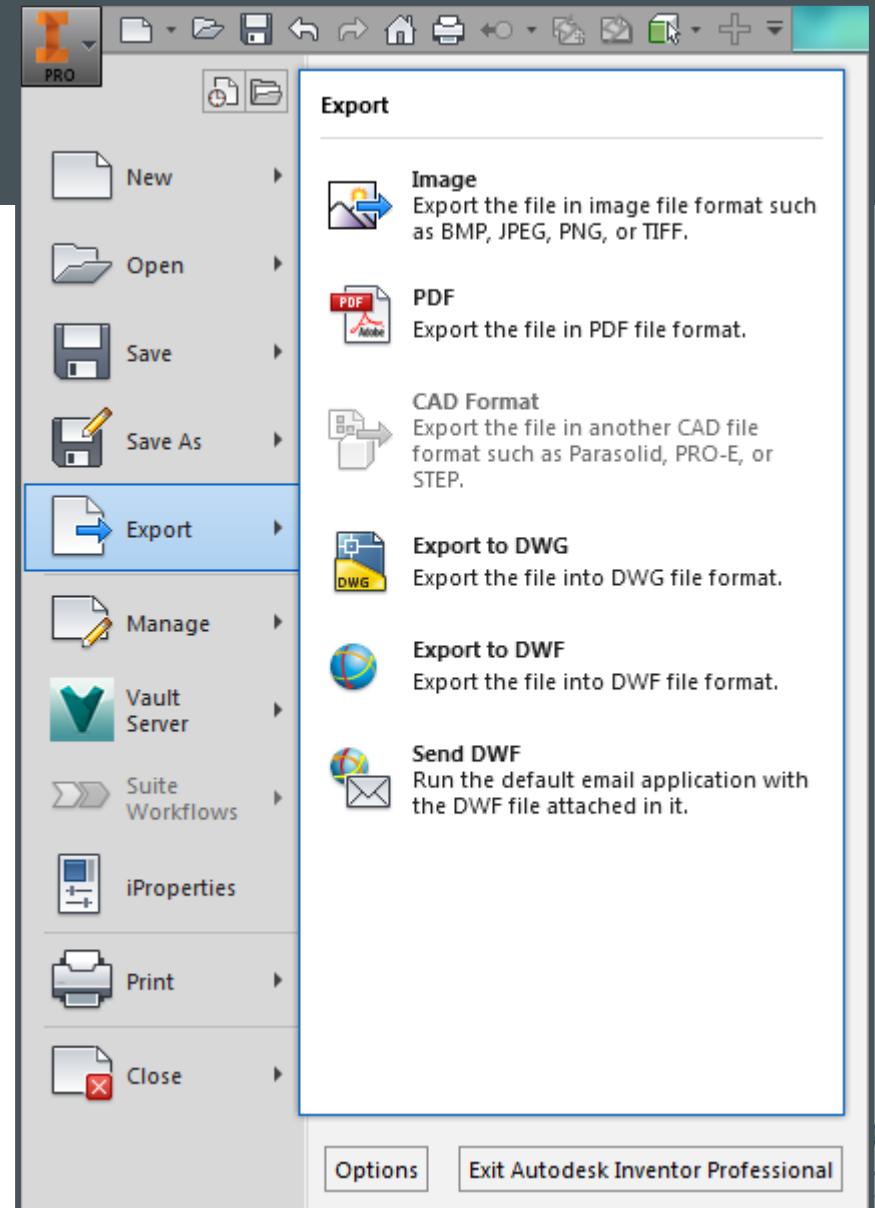
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CHECKED		Piston_Example		
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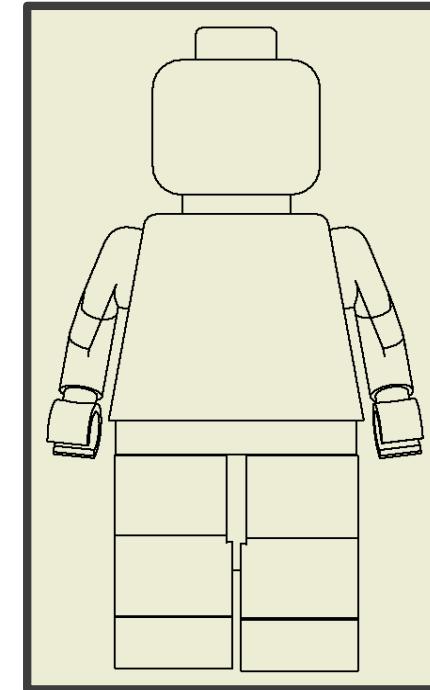
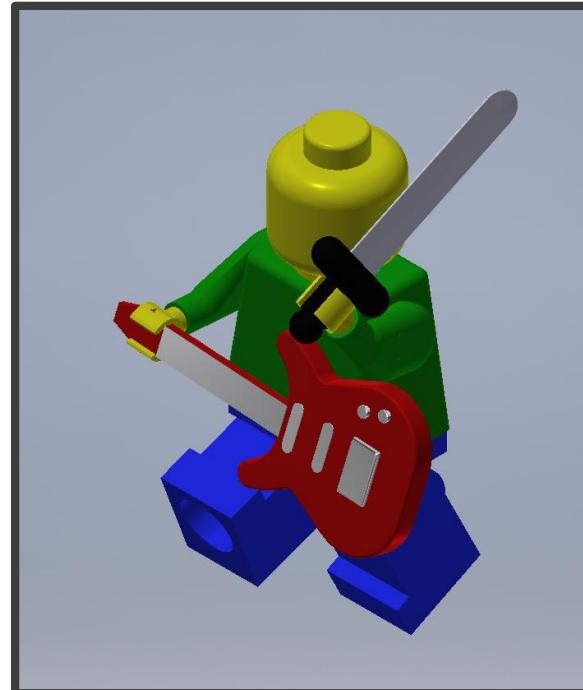
# EXPORT FILE

- Under the Inventor icon, select the **export** option.
- Select the required format type:
  - Images are .jpeg, .png
  - PDF documents are .pdf
  - Drawing files are .dwg
- Name the file and ‘save as’ in the window that appears



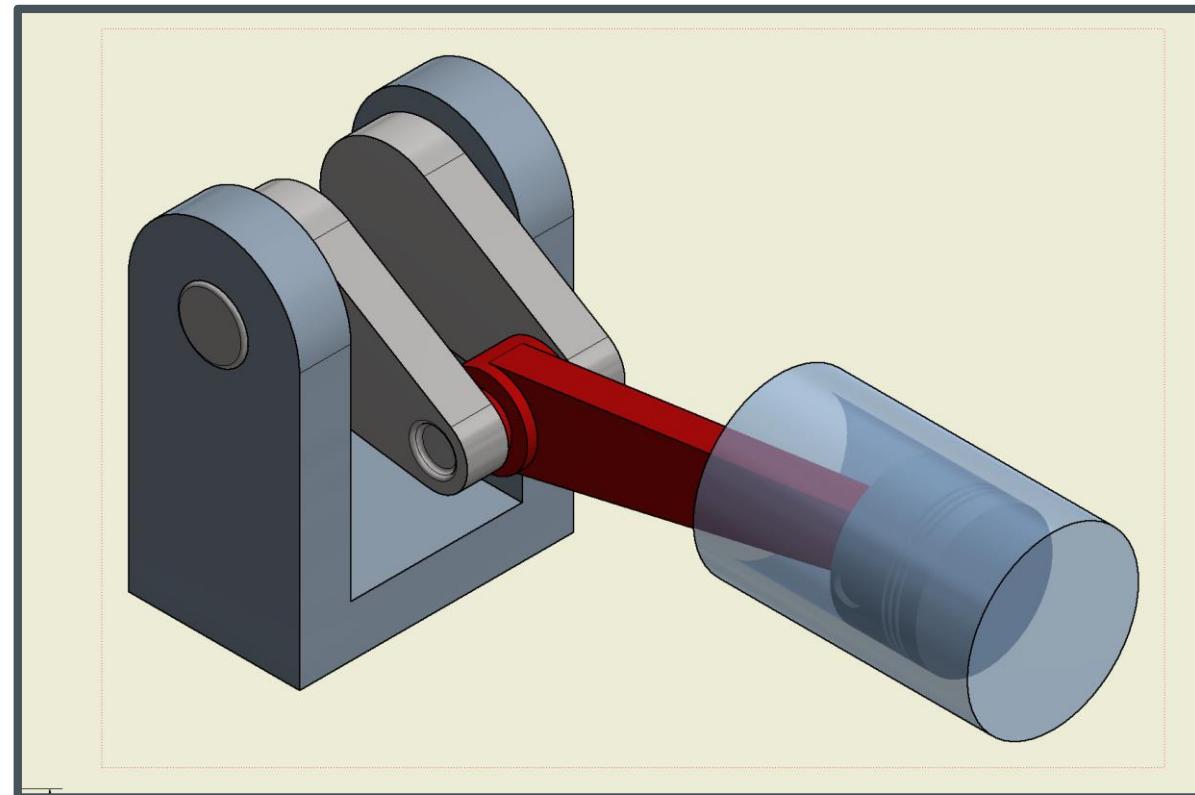
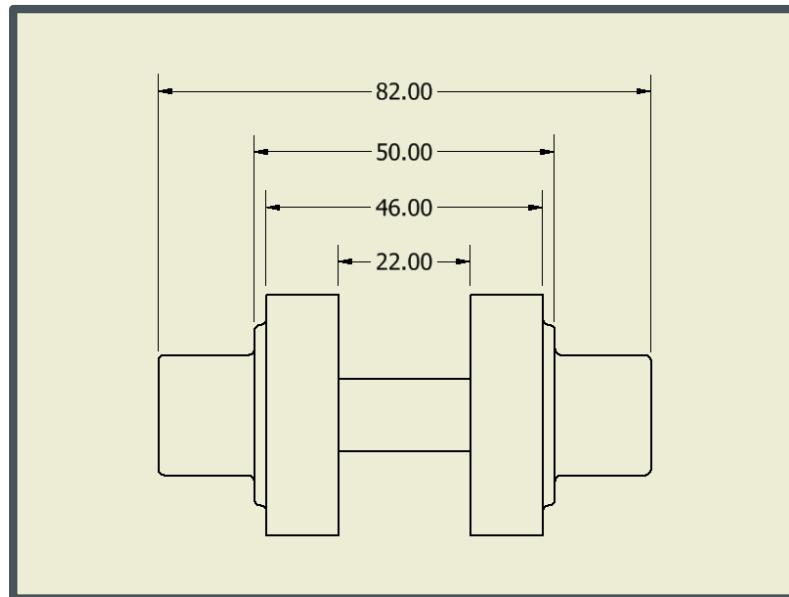
# PRACTICE EXAMPLE 8: LEGO MAN

- Assemble the Lego model and produce drawings of each component and a drawing of the assembly with a parts list included, also try to simulate motion in the arms and legs.
- *All part files can be found in “Lego\_Man” project folder of the lesson materials.*



# DEMONSTRATION: DRAWINGS

- Using the “*piston\_example*” project files provided create a drawing for: 1) each part (*fully annotated*), and 2) the completed assembly (*with a parts list indicating each component*).





# STRESS ANALYSIS

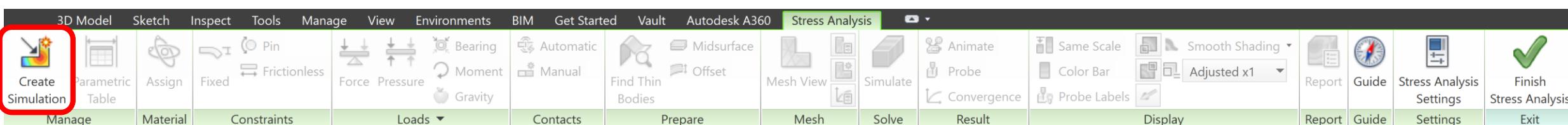
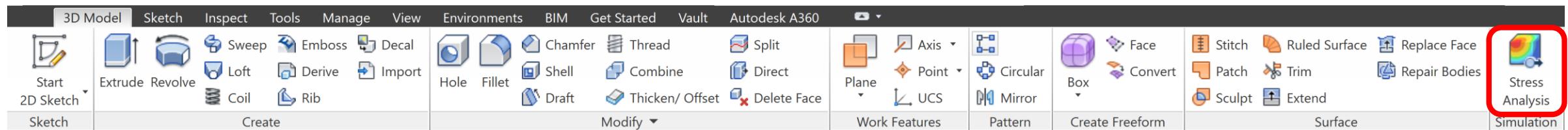
**LESSON 5**



# THE STRESS ANALYSIS MODULE

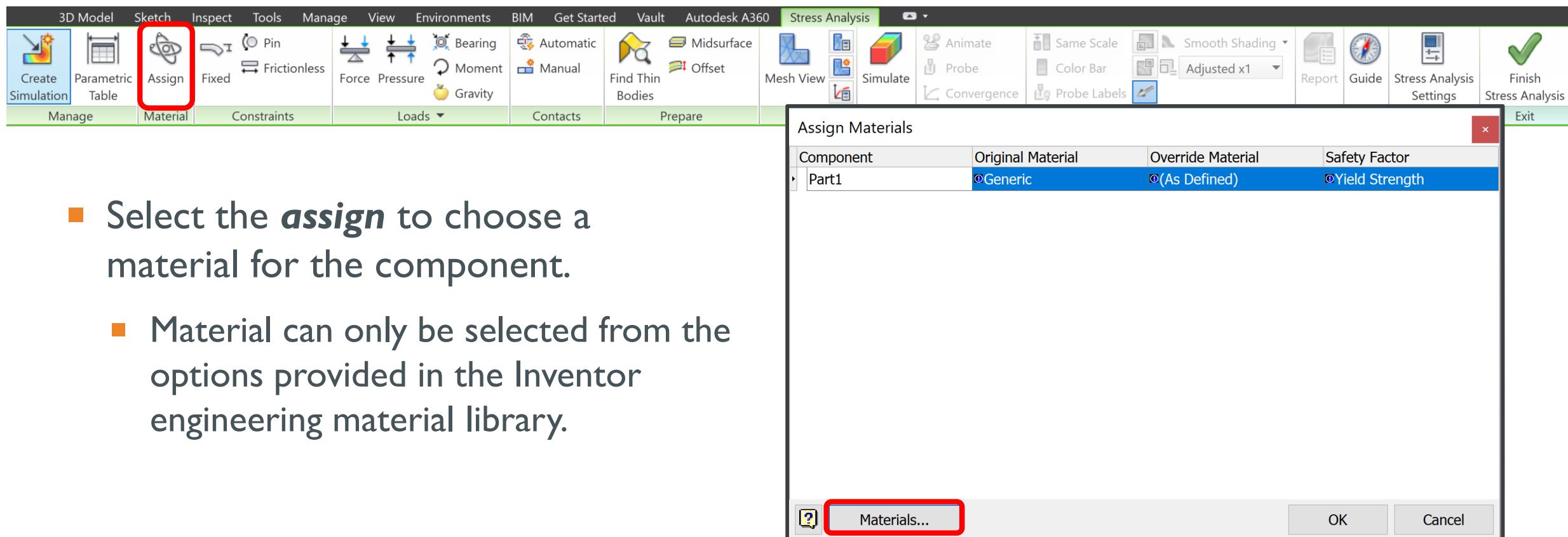
- Inventor has an inbuild stress analysis feature that helps to remove over-engineering in designs by determining where regions of critical stress and fail points occur.
- The general process for analysing a component is:
  - Start a stress analysis
  - Defining the parts material
  - Apply required boundary conditions and input forces
  - Check the component mesh
  - Run the analysis
- **Note:** the Inventor stress analysis module takes a simplistic computational method, would recommend other software packages that are dedicated to finite-element analysis more reliable results.

# START A STRESS ANALYSIS



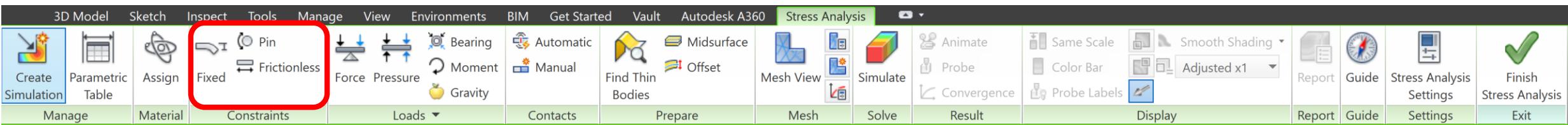
- To start a new stress analysis ensure that the part or assembly that for analysis is present in the graphics window, then select the command from the top ribbon.
- A new command ribbon will open for the stress analysis feature, then select ***create simulation***.

# ASSIGN MATERIAL



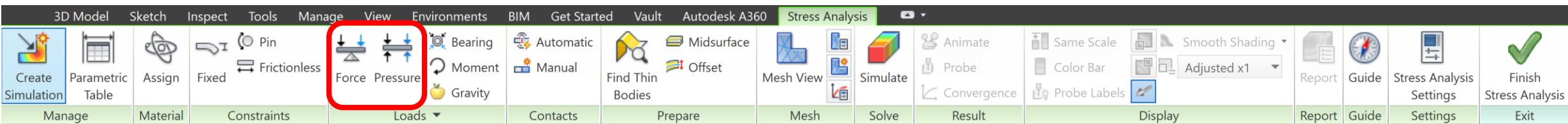
- Select the **assign** to choose a material for the component.
  - Material can only be selected from the options provided in the Inventor engineering material library.

# BOUNDARY CONDITIONS



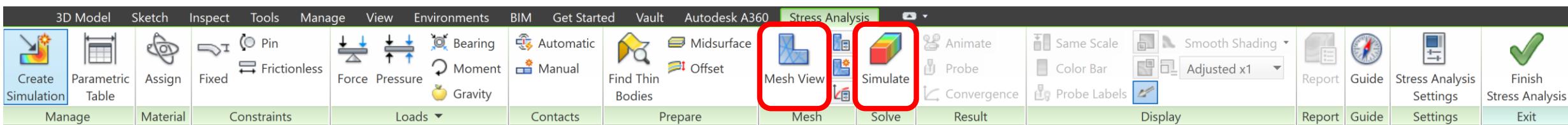
- Apply constraints to the components that will serve as boundary conditions.
  - Constraints can be applied to a single face, or edge which will impact the analysis results.

# INPUT FORCES



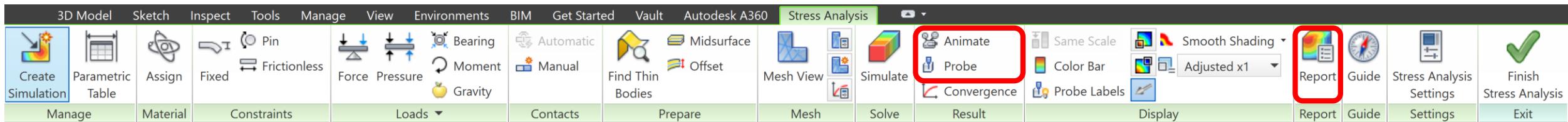
- Apply the input forces which can be applied as a point load or pressure (force distributed over an area).

# MESH VIEW AND SIMULATE



- Select the **mesh view** command to show the meshing for the component in the graphics window.
  - Changes can be made to the mesh relating to the element size which would produce more accurate results, but cannot change the mesh type (shape), which would restrict the module to being more suitable for simple components.
- Select the **simulate** command to begin the stress analysis simulation.

# SIMULATION RESULTS



- Once the simulation has finished generating, the component will be contoured with a range of varying colours: **red** is regions of high stress, and **blue** is regions of low stress.
- A **probe** can be placed on the component to query the stress values in a exact spot.
  - Query results can be turned on an off through **probe label** command.
- The **animation** command will simulate a play-through of the stress application
- To review the analysis results select the **report** command to generate a full report of the analysis variables and simulation results.

# DEMONSTRATION: STRESS ANALYSIS

- Perform a stress analysis and produce a report on a **cantilever beam** (Thursday), or a **simple-loaded beam** (Friday).

