

Computer Aided Design**Introduction to Inventor and feature based modelling**Overview

Feature based modelling systems like Inventor, SolidWorks, CATIA, NX and ProEngineer (now Creo) are true 3D modelling systems that also have the ability to produce associative drawings.

The starting point for a 3D model is a 2D sketch giving a rough idea of form. The sketch can be constrained to ensure that, for example, one line remains perpendicular or parallel to another. Modelling dimensions can then be added to control size.

The next stage is to extrude, revolve, sweep or loft the 2D profile to form a 3D object. Features such as fillets and chamfers can be added. Solids can be shelled to form hollow objects.

All stages of the construction are recorded in a history tree or browser and can be subsequently edited. Feature based models are parametric. The parameters used for shape, size, extrusion depth etc. can be changed and variations of the model created. Each solid operation such as extrusion can be edited as a feature. The sketch from which it was created can be edited and its feature (e.g. extrusion) will be updated accordingly. Parameters can be read in from data tables or spreadsheets with named sizes and equations or dependencies, e.g. length, width=length/2.

Advanced modelling systems like Inventor allow assemblies to be created. Individual parts can be assembled and constrained by relationships such as mating faces, aligned axes and insertions. Exploded assemblies that return to their assembled state can be created and movement can be limited within pre-set ranges or by collision with other parts.

At the part or assembly stage, materials can be added to produce realistic images, or simply to differentiate parts in an assembly.

Drawings are created by arranging views of the model on a sheet and may be orthographic, isometric, auxiliary or sectional projections. Drawings are associative, in that changes to the model are automatically updated on the drawing(s). In most cases, the reverse is possible - a drawing can be modified and the model will update automatically.

File types

Inventor uses the following filename extensions for different file types:

- .ipt** part file
- .iam** assembly file
- .idw** drawing
- .ipn** presentation file

These are the main file types, but in addition Inventor uses a **project file (.ipj)** to keep a record of all the files (parts, drawings, etc.) used in a project, which may contain multiple assemblies. Projects allow for workgroups so that more than one person can contribute to the overall project.

If presentation files are animated, an **.avi** (or **.wmv**) file can be created to record the motion and subsequently be replayed without Inventor.

Sketching

Modelling usually begins with the creation of a 2D sketch. Initial dimensions are unimportant - just a general shape is required. Whilst sketching, constraints that make lines parallel, perpendicular or arcs tangential for example, may be automatically applied. It's a good strategy to make the sketch roughly the right size otherwise applying constraints and dimensions can have a dramatic effect on the geometry!

To create more sketches, click the **Create 2D Sketch** button on the ribbon **Model** tab. When complete, click the **Finish Sketch** button.

Sketches may later be displayed or hidden by selecting **visibility** in the browser.

Sketch Constraints

Additional constraints can be added to control the relationship between sketched lines and arcs:-

Coincident	Collinear	Concentric	Fix
Parallel	Perpendicular	Horizontal	Vertical
Tangent	Smooth (G2)	Symmetric	Equal

To display constraints, right click in the sketch window and click **show all constraints**. By selecting individual constraint symbols, one can see which entities are constrained to each other. Constraints may be deleted. When done, right click and select **hide all constraints**.

Dimensions

Dimensions are applied to sketches to control size. They also constrain the sketch and can lead to over constraining issues! Be careful to decide what is important, shape or size and don't feel you have to constrain everything.

After placing a dimension, if a dialog does not appear automatically, click it and enter the required value.

Dimensions may be hidden by right clicking the sketch in the browser and removing the tick from **dimension visibility**. Do not delete them.

Work planes

Work planes can be used as a platform for sketches where a convenient face is not available. They are created easily on the end of a work axis, but another method is to pick a line in an existing sketch, then in the browser, expand **Origin** and select the desired plane, e.g. XY plane, and enter the angle to this plane.

Work planes can be offset from existing ones by clicking and dragging the existing one and entering an offset distance.

To start Inventor

Click the **start** button and select:-

1. All Programs
2. Autodesk
3. Autodesk Inventor 2018
4. Autodesk Inventor Professional 2018 - English

Inventor tutorials

During your first session you will see a demonstration to get you up and running. In your own time, do some of the online tutorials in Inventor 2018. To run click the **Learning Path** button on the **Get Started** tab. For specific exercises, choose **Help, Tutorials, Tutorial Archive, Autodesk Inventor Tutorials** and then, e.g. **Presentations** (the files for this example can be found at *Filestore on ads\Engineering\TeachingLabs\data\Inventor Tutorial Files\Cylinder Clamp* – Please copy this and any other folders to your own drive, do not attempt to overwrite the sample files). Alternatively, a “Quick guide to presentation files” is available on Blackboard.

Student software

On your own computer you may download a free student copy of Inventor from the Autodesk website (make sure you select the 2018 version - **versions are not backwardly compatible**). Go to www.autodesk.co.uk, then:-

1. **Resources – FREE SOFTWARE FOR STUDENTS (US site) – Get free software**
2. **All products – Inventor Professional** (in the A-Z listings)

Next, click **Create Account** and fill in the form giving your University email address (ending .ac.uk). Once your account and log in details are established, you can return to step **2**, sign in and progress to the download. You will be given a serial no. and a product key – note both of these down carefully as you may need them later.

Project files

The project file (.ipj) maintains links between your parts, assemblies, drawings etc. and also libraries that you may want to access. For each "product", e.g. a complete assembly, create a separate project file, even if you only have one part in mind to begin with.

- Select the **Projects** button on the **Get Started** panel.
- Select **New, New Single User Project** (DO NOT select New Vault Project).
- Right click in the **Project name/location** section and select **New**.
- Give the project a name and under **Project (Workspace) folder** create a new folder under your home folder on the University server.
- Click **Finish**.

Make sure your new project is active (ticked). Any new files that you create should belong to this project.

It is important to work within your project, otherwise Inventor may not be able to find parts, drawings etc. When sending part, assembly or drawing files electronically for marking, send the project file as well.

Libraries

Inventor provides a library of standard parts (fasteners, etc), by default, which you can **Place from Content Centre** when in an assembly file.

Suggested reading

Autodesk Inventor 2018 Essentials Plus, *Banach, D. Jones, T* – SDC Publications, ISBN 9781630570897