

Terminology

You must have a generic understanding of commands and techniques which are used in a number of packages (**YOU CANNOT BE SOFTWARE SPECIFIC**)

You **MUST** use the correct terminology in the exam, you **WILL NOT** be given marks in an exam for using software specific terminology.

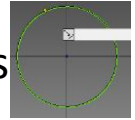
Drawing Tools



- The **LINE** tool creates lines within a drawing.



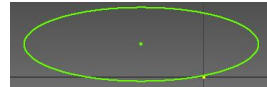
- The **CIRCLE** tool creates a circle either using a centre point and radius



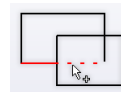
- The **RECTANGLE** tool creates a rectangle.



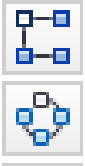
- The **ELLIPSE** tool creates an ellipse using a centre point and major and minor axis.



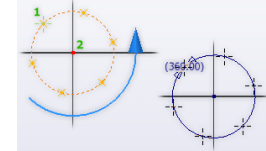
- The **TRIM** tool trims a curve or straight line to the nearest intersecting curve or selected boundary.



Drawing Tools



- The **ARRAY** tool (Linear ,Radial and Rectangular): Duplicates selected sketch geometry and arranges it in rows and columns. Or in an arc or circle in the case of the radial array. Can also be used in a 3D model context



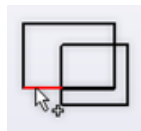
- The **OFFSET** tool duplicates selected sketch geometry and dynamically offsets it from the original.



- The **MIRROR** tool creates a mirrored copy of a sketch across an axis. Can also be used in a 3D model context.



- The **EXTEND** tool extends a curve or line to the next intersecting line or curve.



Drawing Tools

PROFILE: The name given to a 2D shape, prior to being used to make a 3D feature.

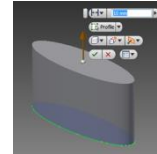
WORKPLANE: A surface where sketches can be applied. Most CAD packages will provide three (elevation, end elevation and plan), but more can be added by the user.

Modelling Features

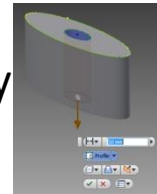


Extrude

- EXTRUDE (Add) creates a feature or body by adding material.

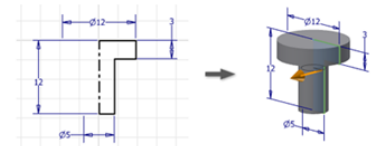


- EXTRUDE (Subtract) removes material from a part or assembly



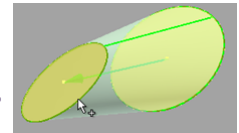
Revolve

- REVOLVE creates a body by revolving one or more profiles around a given axis.



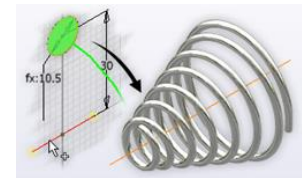
Loft

- LOFT creates a transitional shape between two or more sketches.



Coil

- HELIX creates a helical spring or thread feature. Pitch and revolution or pitch and height must be determined.

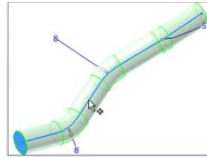


Modelling Features



Sweep

SWEEP sweeps one or more sketch profiles along a path to create a feature or body.

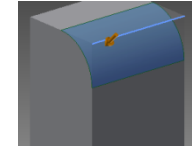


Modelling Edits



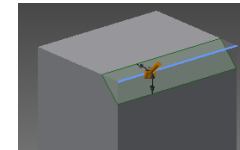
Fillet

FILLET adds fillets or rounds to one or more edges or faces. Can also be used in the 2D drawing tools.



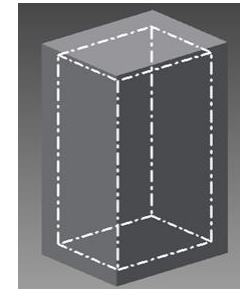
Chamfer

CHAMFER adds fillets or rounds to one or more edges or faces. Can also be used in 2D drawing tools



Shell

SHELL Removes material from a part interior, creating a hollow cavity with walls of a specified size.



Modelling Terms

PROJECTED GEOMETRY: To select a shape from a CAD model or feature and generate it as a new shape in a sketch.

EDGE: The edges of a 3D object.

FACE: The face of a 3D object.

MODELLING TREE: The linear hierarchy of how a 3D CAD model is created or assembled.

ASSEMBLY: An assembly of separate components.

SUB ASSEMBLY: An assembly of components that is added to another, larger assembly.

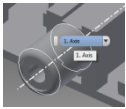
Constraints



MATE: To join the face of a 3D model to another face.



ALIGN: To align the face of a 3D model with another face.



CENTRE AXIS: To find the centres of cylindrical 3D CAD models and align them.



COMPONENT: A single component part, used to create an assembly later on.



SKETCH: The name given to the CAD drawing feature used to create a profile.

CAD LIBRARY: A directory of commonly used parts.

Constraints



LINEAR: causes lines, ellipse axis, or pairs of points to lie parallel to the Y axis.



LINEAR: causes lines, ellipse axis, or pairs of points to lie parallel to the X axis.



CONCENTRIC: causes two or more circles, arcs or ellipses to have the same



PARALLEL: causes selected linear geometry to lie parallel to each other.



PERPENDICULAR: causes selected linear geometry to lie at right angles to each other.

Constraints



TANGENT: Constrains curves and end of a spline to be tangent to other curves.



FIXED: Fixes points and curves in position relative to the sketch coordinate position.

Model Types

SOLID MODEL: Made up of solid shapes and materials added or subtracted as necessary.

SURFACE MODEL: The construction of surface model is done with the use of geometric entities like surfaces and curves. Surface Modelling uses splines and Bezier mathematical techniques for controlling curves

WIRE FRAME MODEL: A wireframe model represents the shape of a solid object with its characteristic lines and points

Modelling Terms

BOTTOM UP MODELLING: Bottom up modeling is when parts are modeled then inserted and fixed in relation to other components in an assembly using mates

TOP DOWN MODELLING: Top down design is an option to create new parts within assembly. Constraints are the existing geometry elements from other parts within assembly: lines, planes, surfaces, points, vertices.

LIGHT SOURCE: The source of light to illuminate a 3D CAD model and scene.

REFLECTION: A reflection in material, colour or light on the 3D CAD model or in the scene.

File Types

STEP FILES: *STEP* file is a CAD file format, usually used to share 3D models between users with different CAD systems.

DXF FILES: DXF stands for **Drawing Exchange Format**. Files that contain the .dxf file extension contain CAD vector image files. DXF file format was developed as an exchange format for the CAD files that are created by computer aided drafting software applications.

3DS FILES: Is one of the file formats used by the Autodesk *3ds* Max 3D modelling, animation and rendering software.