# 3D Modelling a Set of Stairs

## Description

In this activity students will begin by drawing a set of stairs in 2D similar to the one drawn previously using the isometric view. Then they will extrude the drawing to create a three- dimensional rendering. The intent of this activity is to learn how to draw a 2D sketch using the drawing tools and how the constraints and dimensions tools affect how and what you draw.

Drawing methods will vary slightly in each software package, but the overall concepts are the same. As this activity does not necessarily have to follow the previous CAD activities under Mechanical CAD, the student may have less knowledge of the terminology, the drawing tools and how to use them.

## Lesson Objectives

The student will be able to:

* Create a 2D sketch using constraints and dimensions
* Use various tools to edit the 2D sketch
* Extrude a 2D sketch

## Assumptions

The student will:

* Know how to login to a computer and open up the software
* Understand the working environment of the software and how to navigate it
* Have already created a simple 2D sketch in the software (to be repeated as part of this activity)

## Terminology

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**Application button**: the icon in the top left corner of the screen that contains New, Open, Save, etc.

**Assembly constraints**\*: rules that determine how parts in an assembly are placed relative to other parts in the assembly. Constraints remove degrees of freedom. Assembly constraints

include angle, flush, mate, and tangent. Constraints may be placed between faces of features, part edges, points, inferred axes, and part work features such as planes, axes, and points.



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**Assembly modelling\***: two or more components (parts or subassemblies) considered as a single model. An assembly typically includes multiple components positioned absolutely and relatively (as required) with constraints that define both size and position. Assembly components may include features defined in place in the assembly. Mass and material properties may be inherited from individual part files.

**Browser\***: sometimes called the *model tree*, is the graphical hierarchy showing relationships between geometric elements in parts, assemblies, and drawings. Icons represent sketches, features, constraints, or attributes for each model. Objects are shown in the browser in the order in which they were created. Objects may also be edited, renamed, added, deleted, copied, and moved to a different location in the browser.

**Constraints\***: rules that govern the position, slope, tangency, dimensions, and relationships among sketch geometry or the relative position between parts in an assembly. Geometric constraints control the shapes and relationships among sketch elements or assembly components. Dimensional constraints control size. Applying constraints removes degrees of freedom.

**Drawing**: a 2D representation of a part or assembly. The drawing file type has an .idw extension.

**Extrude\***: a feature created by adding depth to a sketched profile. Feature shape is controlled by profile shape, extrusion extent, and taper angle.

**Fully constrained**: when a 2D sketch or 3D part has had all the degrees of freedom removed and it cannot be freely moved anymore.

**Graphics window\*:** the active modelling area in which sketches, constraints, features, parts, and assemblies are created and edited. In the Graphics window, models can be rotated, zoomed in and out, and view characteristics such as colour, material, and light defined.

**Hard snap**: snap function represented by a green dot that appears when snapping to the endpoint of a sketch line. A hard snap is permanent and cannot be moved.

**Home view**: an isometric view of your model. When the Home button on the ViewCube is pressed, it zooms in and re-orients the model in the isometric view in the Graphics window.

**Marking menu**: when you right mouse click in the Graphics window in various modes (sketch, 3D model) a menu comes up with environment-specific command options arranged in a radial, rather than linear, display.

**Navigation bar**: a toolbar containing various tools to move or view your 2D sketch or 3D part in the Graphics window.

**Origin**: the point where the x, y, and z planes or axes intersect.

**Part**: a group of features and faces that have been combined to create a closed volume that is represented as a 3D object.

**Part modelling environment\***: the environment where you create sketches and by using different commands eventually create a 3D part. In part modelling, you create sketches, use feature commands to create three-dimensional features, and then combine the features to create parts.

**Ribbon**: the palette that extends across the top of the Inventor interface and contains multiple tabs for convenient tool access.

**Shell\***: a parametric feature used most frequently for cast or moulded parts. From a specified face, material is removed from the part, leaving a cavity with walls of a specified thickness. Shells usually have walls of uniform thickness, but individual faces can be selected and their thickness specified. Shell walls can be offset to the inside, outside, or both sides of the part, relative to the original part surfaces.

**Sketch\***: consists of the sketch plane, a coordinate system, 2D curves, and the dimensions and constraints applied to the curves. A sketch may also incorporate construction geometry or reference geometry. Sketches are used to define feature profiles and paths.

**Sketch environment\***: consists of a sketch and sketch commands. The commands control the sketch grid and draw lines, splines, circles, ellipses, arcs, rectangles, polygons, or points.

**Sketch plane**: a planar face or work plane on which the current sketch is created.

**Soft snap**: snap function represented by a yellow dot that is not constrained, and therefore can be moved.

**Status bar\***: a display across the bottom of the active window that indicates the next action that the active command requires. When a 2D sketch or 3D sketch is active, the status bar for sketch displays commands specific to the sketch environment.

**STEP file\***: an international format developed to overcome some of the limitations of current data conversion standards. Files created in other CAD systems can be converted to STEP format and imported into Autodesk Inventor.

**ViewCube**: an interface on the Graphics window that helps switch between standard (front, side, top, etc.) views and isometric views of the model.

**Work plane**: the xy (front view), xz (top view), and yz (side view) planes. Sketches and 3D objects are drawn on these planes.

## Estimated Time

45–60 minutes

## Recommended Number of Students

20, based on *BC Technology Educators’ Best Practice Guide*

## Facilities

Computer lab installed with 3D modelling software (Autodesk Inventor, PTC Creo Parametric, SolidWorks, etc.)

## Tools

Projector with computer and speakers, Internet access

## Materials

Handout for students with instructions

## Resources

Instructional video for teacher and students to follow (Inventor 2013):

* + 11.1: Constraining and Dimensioning 2D Sketches
  + 11.2: Drawing Your Stairs
  + 11.3: Extruding Your Stairs

## Teacher-led Activity

Use a computer with a projector and demonstrate the following:

1. Open the program and explain the environment (Browser, Ribbon, ViewCube, Navigation bar, Application menu).
2. Show how to navigate in the sketch environment or part modelling environment using the mouse (scroll left/right, zoom in/out, pan, orbit), the ViewCube, and Navigation bar tools (Navigation Wheel, Zoom, Pan, Orbit).
3. Start a part in imperial or metric.
4. Create a 2D sketch in the sketch environment.
5. Extrude a 2D sketch into 3D in the part modelling environment.
6. Modify a 2D sketch after it has been created in the sketch environment.
7. Modify a 3D part after it has been created in the part modelling environment

## Student Activity

Students will follow the Student Activity “Drawing 3D Stairs” and/or the video tutorials to complete the assignment.

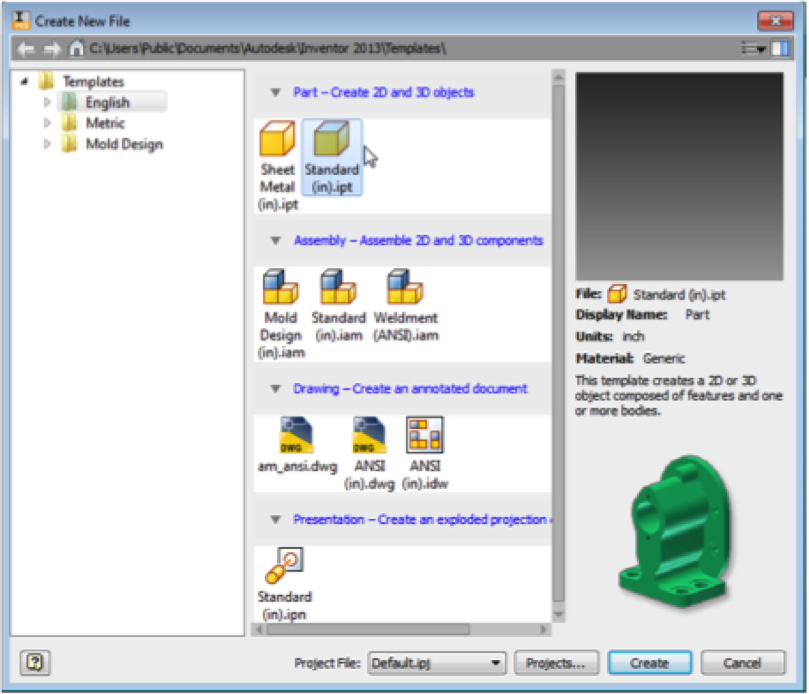
## Assessment

Students will show the teacher their completed assignments. The teacher can have the assignment printed out or look at it on the computer screen. If the student does not produce exactly what was shown, then an associated mark based on errors can be derived.

# Student Activity: Prototyping Your Model Using 3D Printing and CNC Technology

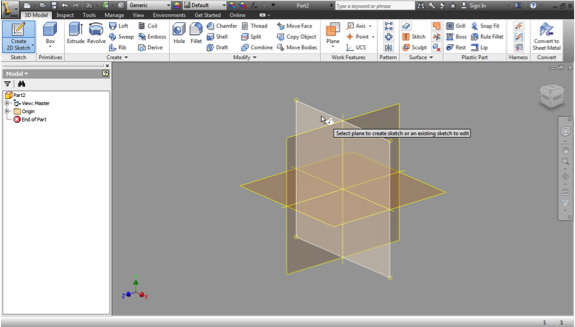
Using the software to learn basic 2D sketch skills, you are going to draw a set of stairs and then extrude them, rendering them as a three-dimensional image (Figure 6).

1. Open a new part from the application menu or from the get started tab on the ribbon (Figure 1).



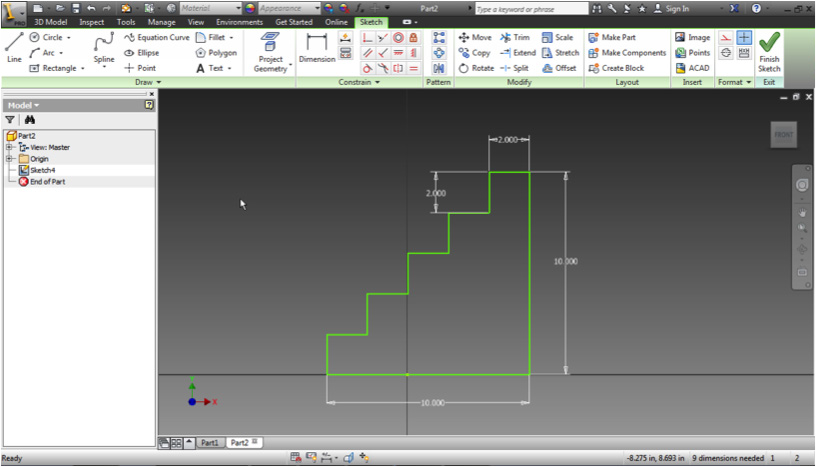
**Figure 1**

1. Next create a 2D sketch and highlight the xy plane (front plane, Figure 2). The program will now enter into the sketch environment.



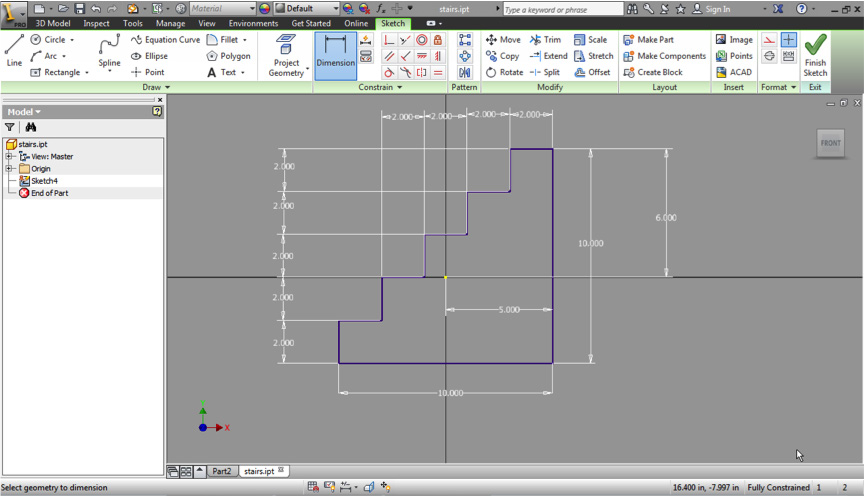
**Figure 2**

1. Draw the stairs as shown (Figure 3). The steps will have a 2" rise and a 2" run. The total height of the stairs will be 10" and the total depth will be 10" as well.



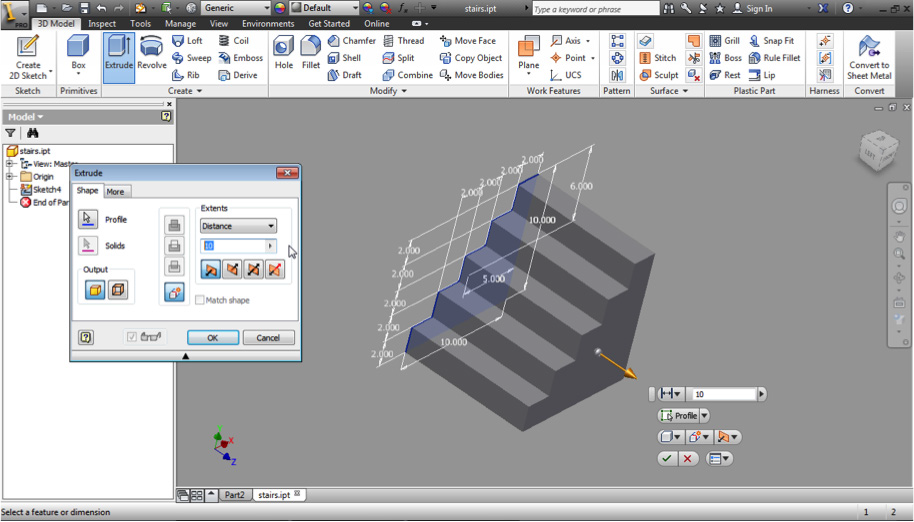
**Figure 3**

1. Dimension the entire drawing so it is fully constrained (Figure 4). In the lower right corner of the status bar it will tell you how many more dimensions you need to fully constrain it.



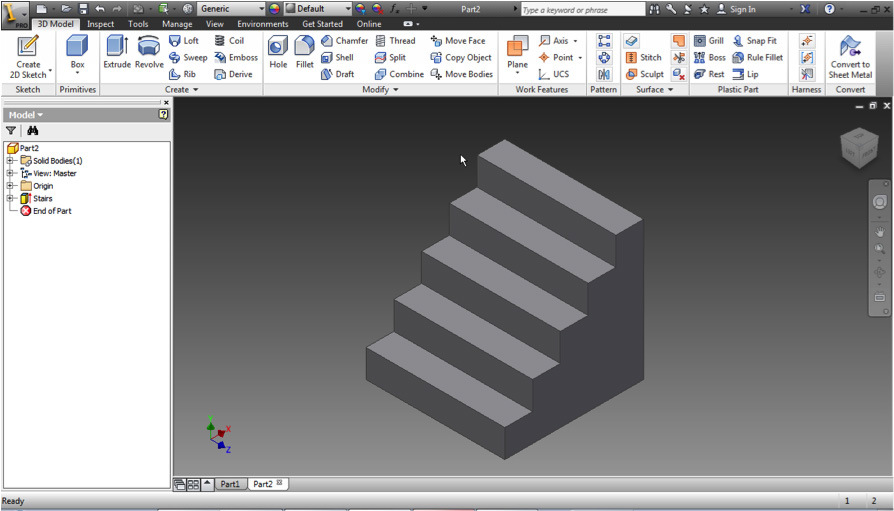
**Figure 4**

1. Once the sketch has been fully constrained, click “Finish Sketch” on the right side of the sketch ribbon to exit the sketch environment.
2. Now select “Extrude” from the 3D model ribbon and select inside the stairs. The stairs will extrude into 3D. In the “Extrude” dialog box, change the depth of the extrusion to 10" (Figure 5). Click OK to finish the extrusion.



**Figure 5**

1. Your model should now be complete and the stairs should look like the ones in Figure 6.



**Figure 6**