**Drawing and Assembling**

**Description**

In this activity the six sides of a die will be drawn and then assembled together. The intent is to understand how constraints are used to “lock” individual parts together to form an assembly.

## Lesson Objectives

The student will be able to:

* Create the six sides of a die
* Draw 2D sketches using constraints and dimensions
* Use various tools to edit the 2D sketch (trim, mirror, offset, rotate)
* Extrude a 2D sketch
* Draw a second 2D sketch and extrude it on the face of an already extruded 2D sketch
* Assemble parts into an assembly

## 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
* Have extruded a 2D sketch into a 3D part in the part-modelling environment

## 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.

**Kerf**: a cut or incision (groove, notch, channel or slit) made in any material by a cutting tool such as a saw, axe or cutting torch.

**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

2–3 hours

## 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):

* + 12.1: Drawing Side 1 of the Die
  + 12.2: Adding a Sketch to a Surface of a Part
  + 12.3: Assembling Your Die

## Teacher-led Activity

Use a computer with a projector and demonstrate the following:

1. Start a new part in imperial.
2. Create the sides of the die in the 2D sketch environment. There are many ways to create the three different sketches needed to complete the die. The suggestion is to start with a 2"

square, then offset and draw the lines for the tabs on one side. Next, mirror, copy, and rotate them to the three other sides. Once on all four sides, use the Trim command to remove the unwanted lines.

1. Extrude a 2D sketch into 3D in the part modelling environment.
2. Save the first created part as another file. A suggestion is to have the students create the first drawing and call it side 1, then “Save As” so that they do not have to restart from scratch to complete the other five sides required.
3. Create a second sketch and extrusion on a sketch that has already been extruded.
4. Assemble the sketches together

## Student Activity

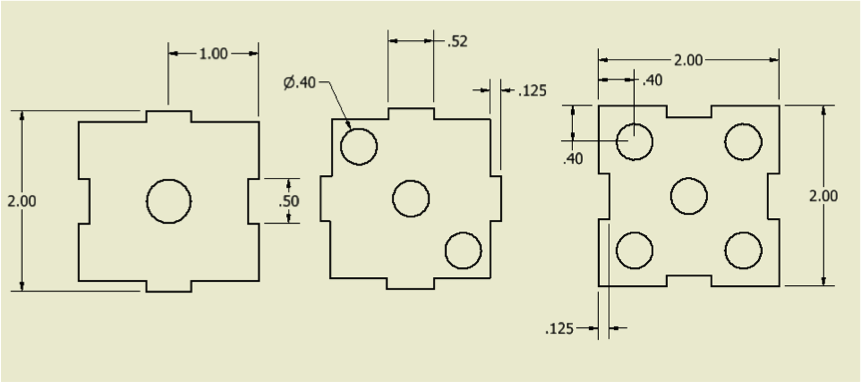
Students will follow the Student Activity “Make a 3D Die” and/or the video tutorials to make their own 3D die.

## Extension Activity

Use the drawing below (Figure 1) for outputting to the laser engraver. Insert this drawing into the Student Activity that follows. Have the students change the tabs from .5" to .52" to allow for the “kerf” of the laser. The kerf is based on ⅛" MDF or ⅛" Baltic birch and may vary depending on the laser engraver you have.

### Note: The male tab will be .02 larger, so that when the die is laser cut it will fit snugly

**without any glue.**



**Figure 1**

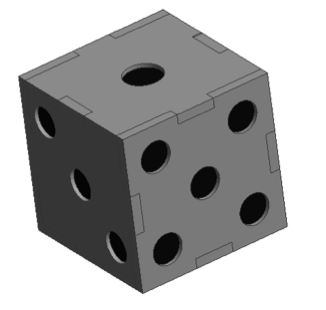
## Assessment

Students will show the teacher their completed assignment. 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 mistakes can be derived.

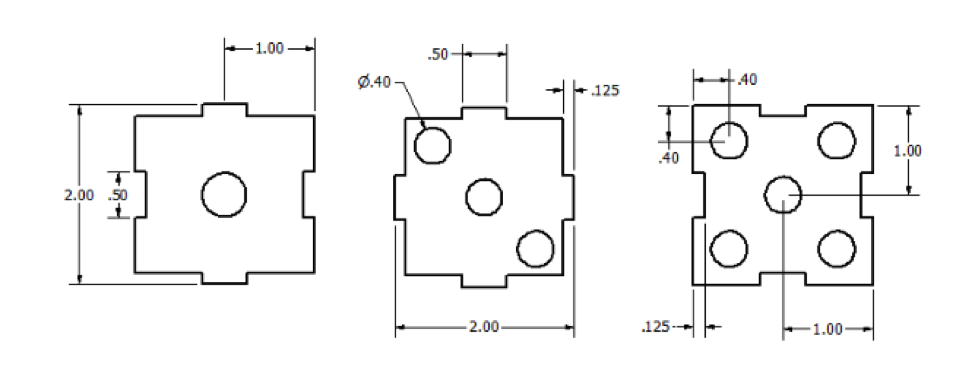
# Student Activity: Make a 3D Die

Using the software, you are to draw a single die that you will assemble on the screen. The dimensions of the die will be 2" cubed. The die will consist of six separate pieces. However, the top/bottom (1 and 6), two sides (3 and 4), and two ends (2 and 5) will each be the same size (Figure 2).

Figure 3 shows the dimensions of the three pieces you will need to create. The thickness of the material will be ⅛". After the three pieces have been created, you will save a copy of each and then change the number of dots to represent the missing sides of the die.



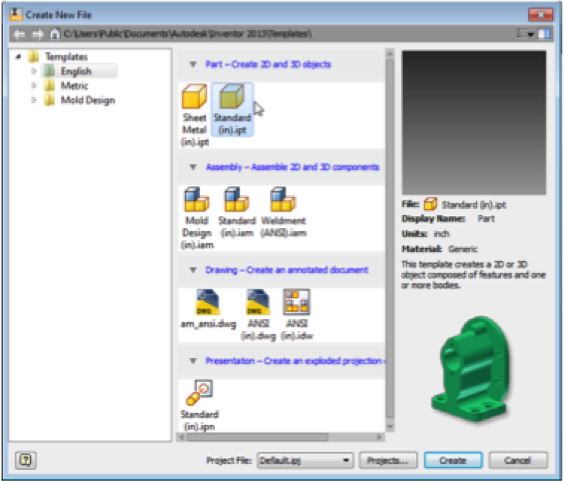
**Figure 2**



**Figure 3**

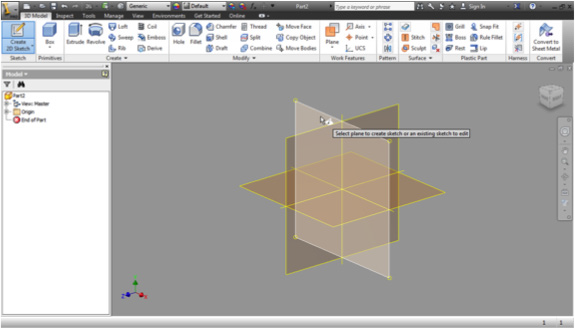
## Procedure for Creating the Sides of Your Die

1. Open a new imperial part from the Application menu or from the Get Started tab on the ribbon (Figure 4).



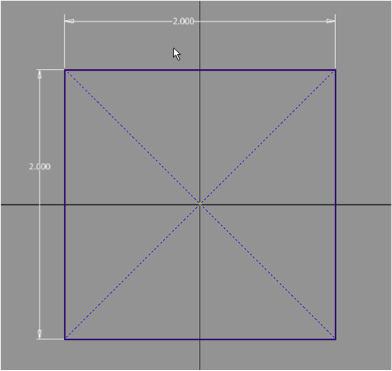
**Figure 4**

1. Create a 2D sketch and highlight ANY plane. The program will now enter into the sketch environment (Figure 5).



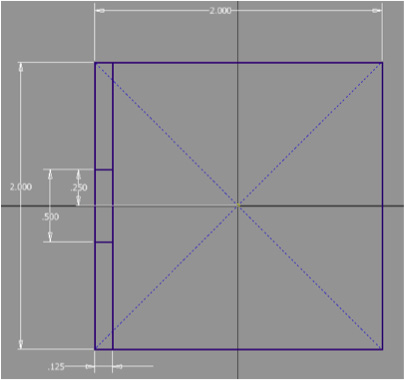
**Figure 5**

1. In the sketch environment, create a 2" by 2" square using the Rectangle command (Figure 6).



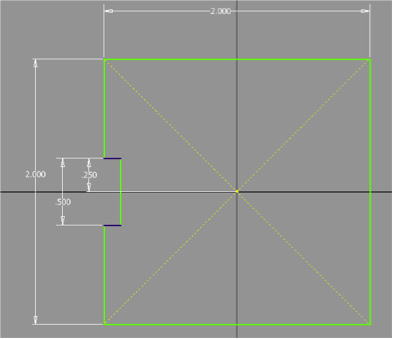
**Figure 6**

1. Offset or draw a line that is .125" in from one of the edges. Draw the start of a tab by drawing 2 lines that are .25" either way from the centre line as shown in Figure 7.



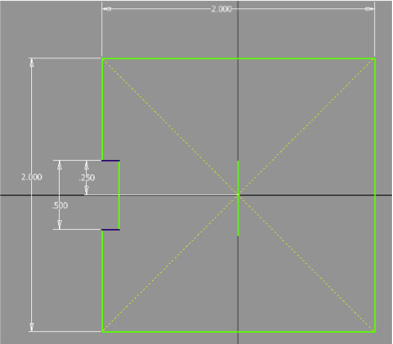
**Figure 7**

1. Trim out the excess lines that are not needed to form the “female” part where a tab will join later (Figure 8).



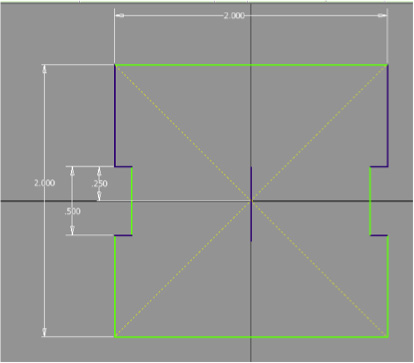
**Figure 8**

1. Add a mirror line on the y-axis so you will be able to mirror the female tab to the other side (Figures 9 and 10).



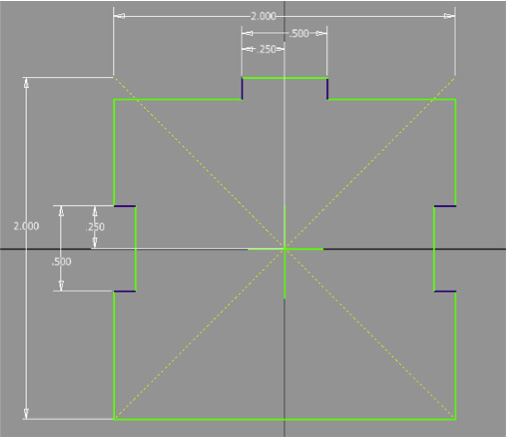
**Figure 9**

1. Using the Mirror command, select the parts you want to mirror and then select the mirror line. The end result will be that the tab ends up on the other side. Trim out the excess to make it look like Figure 10.



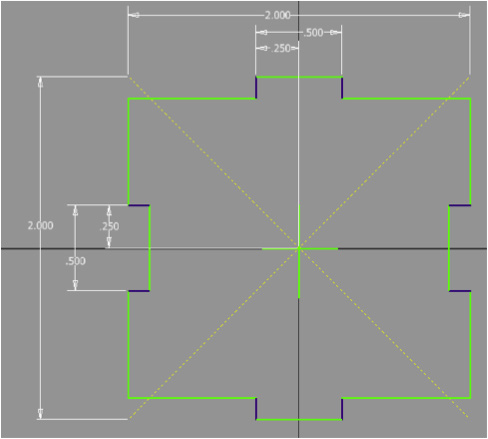
**Figure 10**

1. Using the knowledge from step 7, create the “male” tab at the top as shown in Figure 11.



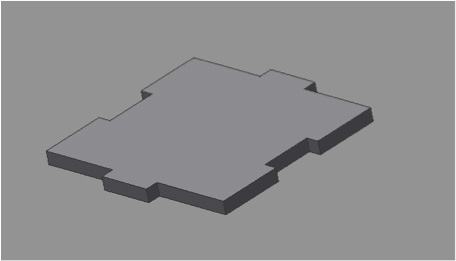
**Figure 11**

1. Mirror and trim out the bottom tab to complete the sketch as shown (Figure 12). Then exit sketch mode.



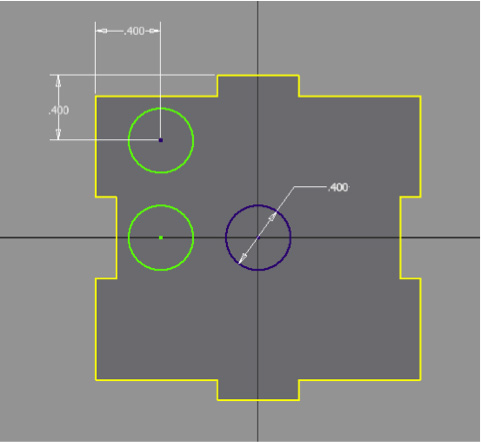
**Figure 12**

1. Extrude the sketch to .125". The end result will be a part looking like the one shown in Figure 13.



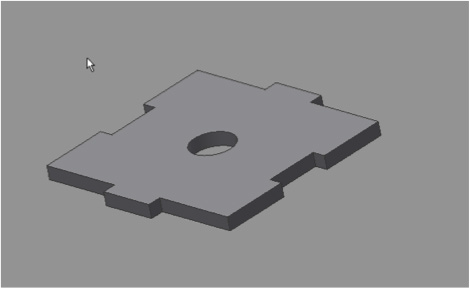
**Figure 13**

1. Add the holes or indents for the dots. To do this, select “Create 2D Sketch” and then select the surface you want. You will enter into sketch mode again. Each dot on the die will be .4" in diameter. They will ALL be .4" from the outside edges or in the centre of the die as shown in Figure 14.



**Figure 14**

1. Once the dot(s) is/are complete for the side of the die you’re working on, finish the sketch (Figure 15). Save your work as *Side 1 – Die*.

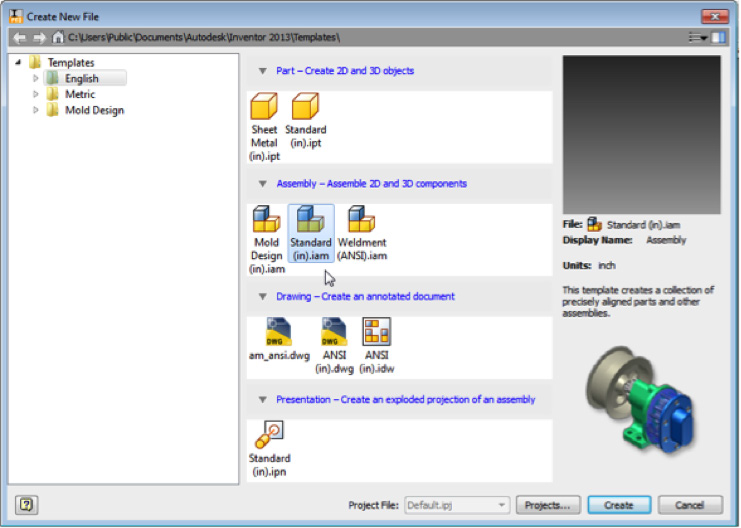


**Figure 15**

1. You have completed one of the sides of the die. Now you must complete the other five sides using what you have learned above. To create the other sides of the die, you can either start a new drawing and copy and paste the sketch from one drawing to another, OR “Save As” the first drawing to start the next side of the die.

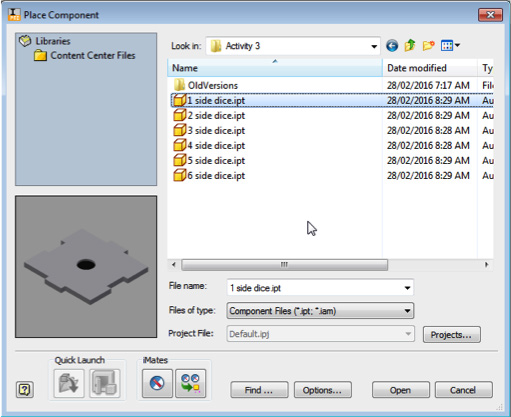
## Procedure for Assembling Your Die

1. On the Application menu, select “Create New File,” and this time choose “Assembly.”



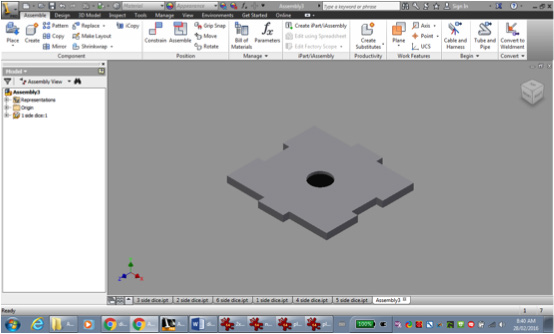
**Figure 16**

1. If you have done everything above correctly, you should have a directory containing the six files listed in Figure 17. When you select a file, the Preview window should show a part that corresponds to the file name you’ve selected (Figure 17).



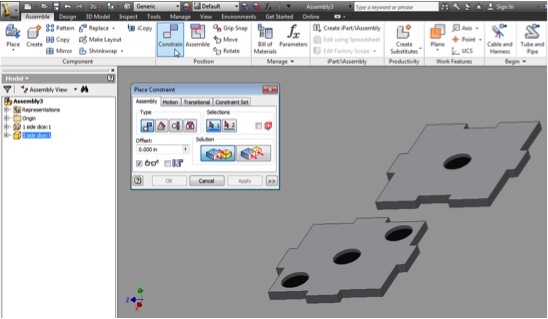
**Figure 17**

1. Place the first part on the screen (Figure 18). This part can be called the grounded part. It is the part that every other piece is connected to. All other pieces will be based off it.



**Figure 18**

1. Place the second part. On the Ribbon, select “Constrain.” The “Place Constraints” dialog box comes up, as shown in Figure 19.

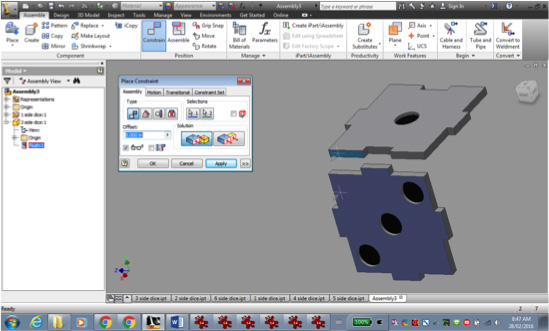


**Figure 19**

Remember which sides are opposite to each other:

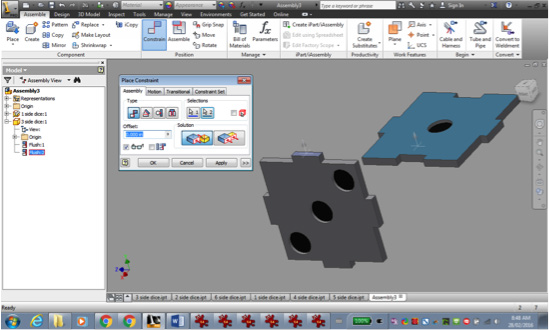
* + 1 is opposite 6
  + 3 is opposite 4
  + 2 is opposite 5

1. Select the two faces of the pieces as shown so that they are aligned flush, as shown in Figure 20. You may have to switch from “Mate” to “Flush” under the “Solution” in the dialog box to get the faces oriented the right way. Select “Apply” when they are lined up correctly.



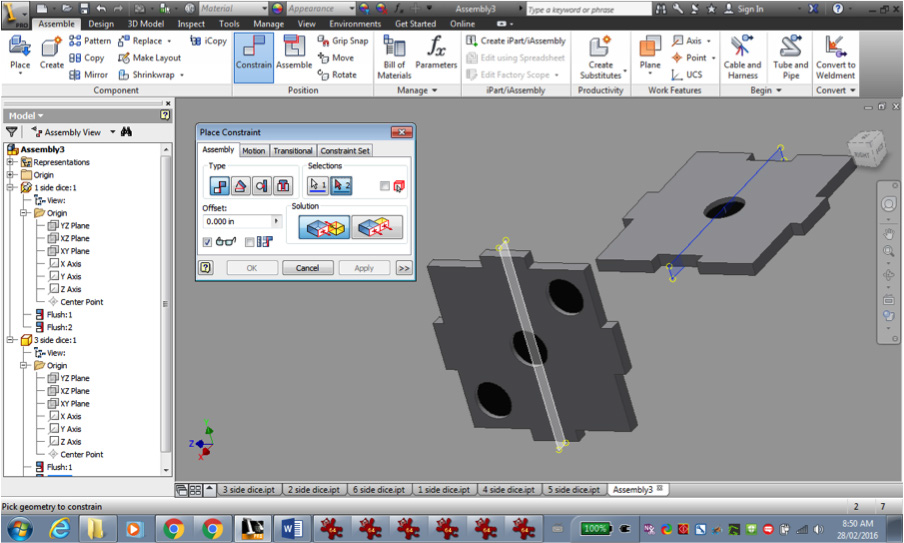
**Figure 20**

1. Select the next two faces of the pieces as shown in Figure 21 so that they are aligned flush. You may have to switch from “Mate” to “Flush” under the “Solution” in the dialog box to get the faces oriented the right way. Select “Apply” when they are lined up correctly.



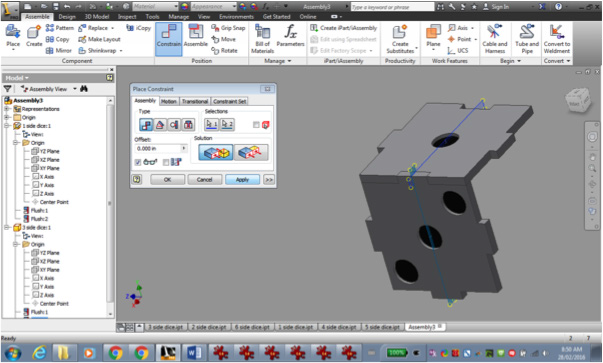
**Figure 21**

1. To ensure the two parts are aligned “Centred,” this time you are going to align them to their planes. In the Browser, expand the parts to show their planes under “Origin.” Picking the planes will highlight them (Figure 22).



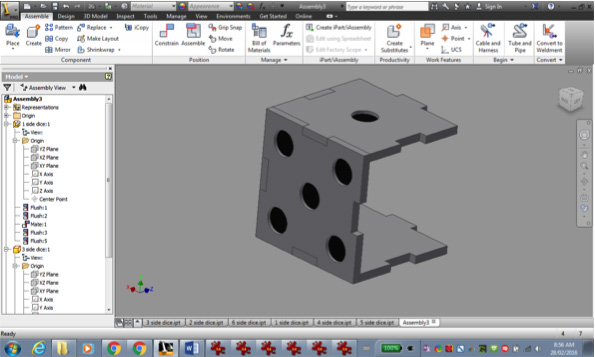
**Figure 22**

1. Once both planes are picked, they will align so that they are mated. Select “Apply” (Figure 23).



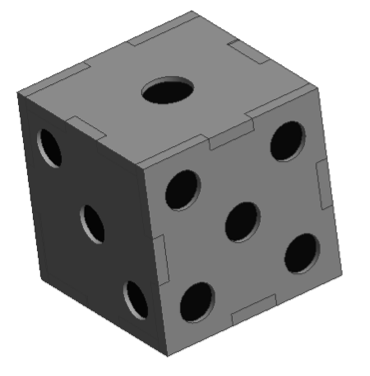
**Figure 23**

1. Continue the process of constraining each part (Figure 24).



**Figure 24**

1. Once you have placed all the parts and constrained them, you should end up with the completed die as shown in Figure 25. Save your work.



**Figure 25**